LINKING CAD AND METROLOGY TO EXPLAIN, DEMONSTRATE, AND TEACH GEOMETRIC DIMENSIONING AND TOLERANCING

BY

KYLE P. HEWERDINE

THESIS

Submitted in partial fulfillment of the requirements
for the degree of Master of Science in Mechanical Engineering
in the Graduate College of the
University of Illinois at Urbana-Champaign, 2010

Urbana, Illinois

Adviser:

W. Brent Hall and James Leake

ABSTRACT

Geometric dimensioning and tolerancing, or GD&T, is a symbolic language that permits design engineers, manufacturing personnel, and quality inspectors to communicate in an efficient and effective manner. This communication focuses on providing a clear definition of geometric features (e.g., surfaces, holes) and the allowable variation that each feature may contain. Unfortunately, owing in part to its complex rule based system, GD&T is also difficult to teach and learn. To address this difficulty, a technique has been developed that allows students to visualize geometric tolerances and tolerance zones, and to directly see when a given data point is in or out of tolerance. The technique employs a portable coordinate measuring machine (CMM) interfaced with parametric solid modeling software, a 3D printer, and a granite surface table to accomplish this. A set of engineering drawings is created, and a 3D printer is used to produce imperfect parts. These imperfections are intended to represent significant manufacturing variation. Then using a portable CMM and the surface table, data points are taken to visually map this manufacturing variation to a 3D parametric modeling software package. Within this software a perfect part is also modeled. Once the inspection data is taken, datum features on the perfect part are used to form the boundaries of the geometric tolerance zones. Through this process, students interactively learn the meaning of datum references, as well as how the various tolerances create different zones. Finally, students use the parametric modeling software to measure the inspection data points to visually see how in or out of specification a given feature is. Having developed a basic working understanding of GD&T, a second module is used to convey design intent through the use of GD&T. Using a simple assembly, students are charged with providing a fully toleranced drawing for one component of this assembly. Students are given a fully dimensioned drawing with basic dimensions and a list of fit, form, and functional

requirements. From these resources, students must choose a datum scheme, tolerance part features, and explain which requirements drive their decisions. In summary, the goal of these educational modules is to illustrate the complex topics of geometric dimensioning and tolerancing through practical application.

I wish to thank my parents, Gerald G. Hewerdine and Lois M. Hewerdine, who conveyed to me at an early age the importance of education. Without the sacrifices they made, I would not have had all the opportunities I have enjoyed. They raised me, supported me, taught me, and loved me. To them I dedicate this thesis.

ACKNOWLEDGMENTS

It is a pleasure to thank the many people who made this thesis possible.

I would like to thank my thesis advisor, Professor Leake, for his willingness to share my vision and his enthusiasm on this project. He devoted countless hours of personal time during the summer, evenings, and weekends to ensure that I was able to complete my degree on time. His level of dedication was above and beyond what was required of him, and I cannot express my gratitude enough.

I am appreciative of Professor W. Brent Hall, Ph.D., for serving as a research advisor.

I am grateful for all the patience of my colleagues at Caterpillar Inc., including Jeremy Wright, Patrick Keeney, Jeff Fullilove, and Michael Lopez. They were always willing to donate time when I had questions. I received as much of an education at Caterpillar Inc. as I did in school.

I would also like to thank Michael Henry for his efforts in peer reviewing portions of this thesis and providing an interview.

A special thank you goes to Eric Gertner for his encouragement, support, and coffee. The advice from his graduate school experience helped to guide me through mine, and his many hours spent reviewing the technical accuracy of the laboratories was vital to the development.

I will always be thankful for the continual friendship and support of my two former roommates, Joe Zearing and Mark Lieber. When I was frustrated, they would always be there with a joke to lighten my mood. I know I can always count on them.

Despite Amanda Sigourney's best attempts, I was still able to focus enough to finish this thesis. Her distractions were always welcome and broke up the chore of writing, and she is a constant bright spot in my life.

I would like to thank my sister Kate Hewerdine for her support.

Finally, I would like to thank my Uncle Paul Wolfrum. As a former University of Illinois graduate from the College of Engineering, he always understood where I was coming from when I would talk about professors, classes, exams, and labs. He kept me focused on the bigger picture of finishing the degree and was a continual source of encouragement.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

Geometric dimensioning and tolerancing (GD&T) is a tool used by engineers and manufacturers to describe part features and their allowable manufacturing variation in an engineering drawing. Through the specification of these properties, designers can guarantee better part quality, part interchangeability, and part function. One company in particular that utilizes GD&T is Caterpillar Inc., where there is a specific group dedicated to analyzing and optimizing the implementation of geometric tolerances [1]. Many of the problems this group encounters originate from engineers lacking sufficient knowledge about GD&T practices. As a result, the company spends significant financial resources on corrective actions due to poor GD&T practices, as well as on teaching engineers the proper fundamentals of GD&T. This is not a problem unique to Caterpillar Inc., and despite the prevalence of the tool across many different industries, often times young engineers do not leave academia with a good foundation in the topic. Engineering programs have some difficulty teaching GD&T due to its complex rule based nature, and the insufficient time available in the curriculum to do the subject justice.

The focus of this thesis, then, is the development of a hands-on, visually based method for teaching geometric dimensioning and tolerancing (GD&T). In order to be successful, the program must provide an ample introduction to GD&T, while presenting it in a stimulating and succinct manner. To attain this goal two laboratory units are developed, one focusing on the fundamentals of GD&T, the other on its application.

In the first laboratory unit a portable coordinate measuring machine (CMM), three-dimensional parametric modeler, granite surface table, and three-dimensional printer are utilized to simulate a manufacturing environment and to perform geometric metrology (i.e., part inspection). The first step is to use the parametric modeler to develop a three-dimensional part with purposely modeled-in variation, and the three-dimensional printer is then used to manufacture this part. The next step in the development involves interfacing the CMM with the parametric modeler, so that geometric metrology can be performed. Teams of students fixture the manufactured part to the granite surface plate, and then inspect the part variation with the CMM. This interactive unit serves to familiarize students with the concepts of datums, manufacturing variation, and geometric tolerances and their boundaries.

The second laboratory unit focuses on applying geometric tolerances to reflect design intent. Students are given a design scenario which includes: 1) an engineering drawing with basic tolerances, 2) a description of the process where the design is used, and 3) a list of acceptable variational requirements that the design must meet. Students then use the process description and the variational requirements to derive tolerances for the drawing, and complete a follow up worksheet to explain which requirements drive the tolerance scheme. Through this exercise students learn how to reflect design intent in an engineering drawing through the use of geometric tolerances, while reinforcing and practicing the principles of GD&T learned during first laboratory unit.

This program aims to not only educate students on the basics of GD&T, but also to motivate student interest in the topic through the use of a hands-on, visually based laboratory exercises. These exercises are designed to be more effective than the customary rule based explanation of GD&T, because they incorporate an interactive component that traditional approaches leave out. Instructors are able to efficiently cover multiple subjects, and students stay engaged during the educational process, as they physically practice GD&T, parametric modeling techniques, and metrology. While this program is not a substitute for a full training class on GD&T, it should provide a sufficient fundamental base for students.

CHAPTER 2

BACKGROUND

Geometric dimensioning and tolerancing, or GD&T, is a symbolic language created to allow design engineers, manufacturing personnel, and quality inspectors to communicate efficiently and effectively. The communication between these parties is focused on providing a clear definition of geometric features such as surfaces and holes, and the allowable variation that each of these features may contain. This symbolic language, like other languages, is continually evolving, and is officially documented in what is known as a standard. There are different standards for different applications, but in general a standard is comprised of a specific set of characters, rules, and definitions which are intended to promote conformity within a specific application. In the case of dimensioning and tolerancing, each standard has its own set of unique marks, symbols, and indicators that are designed to provide a concise, accurate description of the various features to be produced. These symbols are then used in a document known as an engineering drawing, which can be read by others who also understand the language, without any need of additional explanation. The standard this thesis is based on is the American Society of Mechanical Engineers (ASME) Y14.5M-1994 Dimensioning and Tolerancing standard [2]. Many of the concepts discussed in this standard are applied in a similar manner in other geometric dimensioning and tolerancing standards, including the International Organization for Standardization (ISO) series. This series includes 1) ISO 1101:2005 Geometrical Product Specifications (GPS) - Geometrical tolerancing - Tolerancing of form, orientation, location and run-out, 2) ISO 5458:1998 Geometric Product Specifications (GPS) - Geometrical tolerancing - Positional tolerancing, and 3) the ISO 5459:1981 Technical drawings - Geometrical tolerancing - Datums and datum-systems for geometrical tolerances. This method of exchanging information through an engineering drawing is so widely accepted, that it is often used as the basis for legal documents. The ability to read and speak this language is one of the most important skills for an engineer to develop in order to become an effective communicator.

2.1 History of Geometric Dimensioning and Tolerancing

Geometric dimensioning and tolerancing has been around since the early 1900's, but did not gain prominence until large manufacturing volumes became prevalent in the mid 1900's. Previously assemblies were made in such low volumes that incorrectly produced parts could be re-worked, assemblies could be altered, and the quantity of scrapped parts was low enough so as not to have a large financial impact on a company's bottom line. However, with the start of America's involvement in World War II, the production level of warships, aircrafts, and vehicles increased dramatically and the efficient utilization of resources was crucial to America's war effort [3]. Out of specification parts became a huge problem during the war because in such large volumes these errors were not easily remedied and they significantly impacted efficient manufacturing. After the war ended many manufacturing industries, most notably the auto industry, adopted the use of GD&T as a technique to reduce manufacturing errors and increased costs in high volume production. In 1982 the American National Standards Institute (ANSI) rigorously defined and standardized GD&T in the ANSI Y14.5M-1982 standard.

2.2 Use of GD&T in Industry

To better define what purpose geometric dimensioning and tolerancing serves in industry, Caterpillar Inc. employee Michael Henry was interviewed for this thesis [1]. Henry is the team leader for the statistical tolerance analysis (STA) group. The purpose of this group is to resolve all issues involving both GD&T and either manufacturing, fabrication, or new product validation within Caterpillar.

2.2.1 Michael Henry's Formal GD&T Training

Henry studied mechanical engineering at the University of Missouri - Columbia, where only dimensional tolerancing and fit classifications were covered in a few lectures. When Henry graduated in 1991, he began working in manufacturing where he occasionally encountered GD&T. During these projects, GD&T was addressed by referencing published standards such as ASME Y 14.5M-1994. This was done on a case by case basis and no formal GD&T training was needed.

When Henry took a position at Caterpillar within the STA group, he still had no formal training in GD&T. He was sent to an intensive three day training program taught by Technical Consultants Inc [4]. Having established a baseline understanding of GD&T, Henry also went through tolerance stack-up analysis training at the University of Wisconsin-Milwaukee [5], and Sigmetrix CeTOl Six Sigma training, hosted by Sigmetrix [6]. CeTOL is a specific program used for statistical geometric tolerance stack-ups. In these training sessions, Henry learned that his colleagues also had little in the way of formal GD&T training while in school. Through these experiences, as well as in conversations with his peers within Caterpillar, Henry believes that it is common for students to graduate from engineering school with no formal GD&T training.

2.2.2 Effects of Limited GD&T Training on Engineering Drawings

As a consultant group within Caterpillar for geometric dimensioning and tolerancing, Henry's group sees a wide variety of drawings from many different design engineers. Henry states that most of the customers he works with only possess a basic understanding of GD&T and frequently they simply rely on, "The way it was done before." As a result, many of the drawings are only partially toleranced, the datum schemes are incomplete, circular references exist between tolerances, and overall the engineering print does not reflect the fit, form, and function of the final product.

2.2.3 Consequences of Limited GD&T Training on Caterpillar Inc.

These drawing errors can ultimately affect Caterpillar's bottom line. Caterpillar's engineering drawings serve as a legal and binding agreement between the company and their suppliers. As a result, if an engineer is not careful and just carries over tolerances or has an incomplete tolerance scheme, Caterpillar is obligated to accept parts that may have too much variation for their intended function. Alternatively, Caterpillar would also have to reject parts that do not meet the print, even though they could potentially function properly. Both of these scenarios increase the supplier's cost, which ultimately raises the price Caterpillar pays for manufacturing, and adversely affects the relationship between Caterpillar and their suppliers.

Additionally, if an engineer does not understand tolerancing, he/she can artificially make the tolerance value tighter than necessary, and this can lead to increased manufacturing costs. While the part may be manufactured correctly, the level of control required by the drawing increases the complexity and time required to produce the part. The converse is also true. If tolerances are too loose, re-work may be required to correct the manufactured parts, or at the very least extra time must be spent by the engineer approving the out of tolerance part. This too raises manufacturing costs to Caterpillar. In the end, all these factors hinder Caterpillar's ability to produce a competitive product within the heavy equipment, power generation, and construction industries. As a result, groups such as the STA team must exist to help correct these problems.

Extrapolating Caterpillar's Scenario

As mentioned previously, Henry's group uses CeTOL Six Sigma software from Sigmetrix to perform statistical tolerance analysis to optimize the implementation of geometric dimensioning and tolerancing. On Sigmetrix's website it is stated that their product is also utilized in the aerospace, automotive, bio-medical device, consumer products, electronic, and hi-tech electronic industries [7]. Since these industries utilize the same software as Caterpillar, it is natural to assume that they also place considerable emphasis on geometric dimensioning and tolerancing. Therefore it can be assumed that geometric dimensioning and tolerancing is applicable to many disciplines of engineering. Some of the other companies that utilize CeTOL include: ABB, Audi, BAE Systems, Bosch, Chrysler, Cummins, Detroit Diesel, Diebold, Eaton Electrical, Emerson, Ericsson, Fujitsu, General Electric, Goodrich, Hitachi, Honeywell, John Deere, Motorola, Northrop Grumman, Panasonic, Rolex, PACCAR, Polaris, Raytheon, Roche, Sandia Nat'l Labs, Schlumberger, St. Jude Medical, Stanley, Stryker, Tetra Pak, Toyota, TVS Motors, Tyco Electronics, U.S. Army, Volvo, Whirlpool, and Xerox.

2.3 Geometric Dimensioning and Tolerancing in Academia

To gauge the degree to which GD&T is taught in academia, the course descriptions for the top ten mechanical engineering programs in 2010, as ranked by *U.S. News and World Report*, were reviewed [8]. These schools include:

- Massachusetts Institute of Technology
- Stanford University
- University of California – Berkeley
- California Institute of Technology
- University of Michigan - Ann Arbor
- Georgia Institute of Technology
- University of Illinois - Urbana-Champaign
- Cornell University
- Purdue University - West Lafayette
- Princeton University

After reviewing the course descriptions for these programs, the schools were broken down into three categories: 1) those that make no mention of tolerances or GD&T, 2) those that include some material on tolerances, and 3) those schools which specifically mention GD&T.

2.3.1 No GD&T

Of the top ten mechanical engineering schools, University of California - Berkeley [9], California Institute of Technology (Cal Tech) [10], Georgia Institute of Technology (Georgia Tech) [11], and Princeton [12] made no mention of tolerances or geometric tolerances in their course descriptions.

2.3.2 Some Material on Tolerances

Within the top ten mechanical engineering schools, three schools mentioned tolerances as one component of a broad mechanical engineering course.

Massachusetts Institute of Technology

MIT has a course on computational geometry, 2.089J, where variational geometry, tolerances, and inspection methods are specifically covered. The full course description is included below [13].

Topics in surface modeling: b-splines, non-uniform rational b-splines, physically based deformable surfaces, sweeps and generalized cylinders, offsets, blending and filleting surfaces. Non-linear solvers and intersection problems. Solid modeling: constructive solid geometry, boundary representation, non-manifold and mixed-dimension boundary representation models, octrees. Robustness of geometric computations. Interval methods. Finite and boundary element discretization methods for continuum mechanics problems. Scientific visualization. Variational geometry. Tolerances. Inspection methods. Feature representation and recognition. Shape interrogation for design, analysis, and manufacturing. Involves analytical and programming assignments.

Stanford University

Stanford offers two classes which mention tolerances; one is a design course and the other is a precision manufacturing course. The design course, ME 317B, involves creating a design and then developing a manufacturing process for this design. One of the criteria included for this project is that tolerance analysis must be completed on the design. The course description is below [14].

Building on 317A, focus is on the implementation of competitive product design. Student groups apply structured methods to optimize the design of an improved product, and plan for its manufacture, testing, and service. The project deliverable is a comprehensive product and process specification. Topics: concept generation and selection (Pugh's Method), FMEA applied to the manufacturing process, design for robustness, Taguchi Method, SPC and six sigma process, tolerance analysis, flexible manufacturing, product testing, rapid prototyping.

The second course Stanford offers is a precision engineering course, ME 324, where manufacturing, tolerances, and metrology are discussed along with other concepts. The course description is below.

Advances in engineering are often enabled by more accurate control of manufacturing and measuring tolerances. Concepts and technology enable precision such that the ratio of overall dimensions to uncertainty of measurement is large relative to normal engineering practice. Typical application areas: non-spherical optics, computer information storage devices, and manufacturing metrology systems. Application experience through design and manufacture of a precision engineering project, emphasizing the principles of precision engineering. Structured labs; field trips.

Purdue University - West Lafayette

While the University of Purdue has four courses which mention tolerances, there is no specific mention of geometric tolerancing or dimensioning. The course description for ME 11500 - Engineering Drawing I is listed below [15].

A technical drawing course covering engineering geometry, orthographic projection, auxiliary views, dimensioning, and tolerancing using sketching

techniques, and 2-D CAD.

The second course is focused on modeling. The course description for ME 16000 - Solid Modeling is listed below.

Communication of form and layout of real world objects, solid modeling of objects. Engineering drawing layouts, orthogonal projections, dimensioning, tolerancing and standard drawing symbols, principles of detail design drawings and assembly drawings, and manufacturability. Use of computer graphics and production of drawings.

The other two courses offered at Purdue focus on design. The machine design class, ME 46100, focuses more on the fundamentals of design and includes an area on fits and tolerances. The course description is shown below.

Application of mechanics and mechanics of materials to the analysis and design of machine elements. Stress and deflection analysis, statistical considerations under steady and variable loading, stress principles applied to fasteners, springs, welded joints, and general mechanical elements. Fits and tolerances. Antifriction bearings. Spur gears. Introduction to finite element analysis. Laboratory includes projects, solutions of design problems, and experiments.

The second of these design classes focuses on design for manufacturing, where manufacturing process selection and the tolerances associated with each process are covered. The course description of ME 55700 is included below.

Introduction to manufacturing concerns, such as efficient design, producibility, and quality, which must be considered early in the engineering design process. Topics include the product development cycle, manufacturing process selection, tolerancing, quality function deployment (QFD), design for assembly (DFA), quality control techniques, Taguchi's robust design methodology, life cycle engineering, and reliability. Laboratory projects in the area of tolerancing, assembly, and manufacturability are included along with a project from industry in which the students can disassemble, analyze, and redesign a product while obtaining feedback from industry concerning manufacturability.

2.3.3 Schools with Specific Mention of Geometric Dimensioning and Tolerancing

Three schools specifically mentioned geometric dimensioning and tolerancing as a part of their mechanical engineering curriculum.

University of Michigan-Ann Arbor

The University of Michigan offers two courses which include tolerances as a component. The first course is statistical quality control and design, ME 401, which focuses on engineering processes, the tolerances associated with them, and the use of measurements to ensure quality. The course description is included below [16].

Evolution of quality methods. Fundamentals of statistics. Process behavior over time. Concept of statistical process control (SPC). Design and interpretation of control charts. Process capability study. Tolerance. Measurement system analysis. Correlation. Regression analysis. Independent t-test and paired t-test. Design and analysis of two-level factorial experiments. Fractional factorial experiments. Response model building. Taguchi methods. Case studies.

ME 588 focuses is a design and manufacturing course with emphasis placed on assemblies, datums, and GD&T. The course description is below.

Assembly on product and process. Assembly representation. Assembly sequence. Datum flow chain. Geometric Dimensioning & Tolerancing. Tolerance analysis. Tolerance synthesis. Robust design. Fixturing. Joint design and joining methods. Stream of variation. Auto body assembly case studies.

University of Illinois - Urbana-Champaign

The University of Illinois has two courses which incorporate tolerancing and geometric dimensioning [17]. The first class is an introductory computer aided design course, where students learn how to use a parametric modeling software package, create computer based drawings, cover the ISO and ANSI standards for coordinate dimensioning, and are introduced to geometric dimensioning and tolerancing. The course description for ME 170 is listed below.

Geometry and topology of engineered components: creation of engineering models and their presentation in standard 2D blueprint form and as 3D wire-frame and shaded solids; meshed topologies for engineering analysis and tool-path generation for component manufacture; ISO and ANSI standards for coordinate dimensioning and tolerancing, with an introduction to geometric dimensioning and tolerancing. Use of ProEngineer 3D solid-modeling software for creating associative models at the component and assembly levels with automatic blueprint creation, interference checking, and linked bill of materials.

The second course offered at the University of Illinois is a design for manufacturability course, where geometric dimensioning and tolerancing is covered in the inspection and metrology unit. The course description for ME 350 is included below.

Introduction to design-for-manufacturability methodologies and tools; quality management (Taguchi, productivity function deployment, statistical process control, etc.); materials selection (new and traditional materials); designing for primary manufacturing processes (cutting fundamentals, casting, forming, and shaping); designing with plastics (snap-fits, integral hinges, etc.); design for assembly; design for inspection and metrology (datums, geometric tolerancing, and inspection equipment); computer-integrated manufacturing.

Cornell University

Cornell has a specific class dedicated to dimensional tolerancing and mechanical design, MAE 5200. While conventional tolerancing is covered, the main focus of the class is modern GD&T and its role in assembly control. The course description is below [18].

Designers use dimensional tolerances to limit spatial variations in mechanical parts and assemblies; the primary goals are interchangeability in assembly,

performance, and cost. This course covers <u>traditional limit tolerances</u> briefly but focuses mainly on <u>modern geometric tolerances</u> and their role in assembly control. Students learn how to represent assemblies in terms of mating and relational constraints, and how to <u>design tolerances and inspection procedures</u> from part and assembly specifications.

2.4 ASEE Publications

Finally, to gain a better understanding of how schools incorporate GD&T into their engineering courses, the American Society of Engineering Education (ASEE) website was consulted. The ASEE defines itself as a nonprofit organization of individuals and institutions committed to furthering education in engineering and engineering technology, and its members include deans, department heads, faculty members, students, and industry representatives in all disciplines of engineering and engineering technology [19]. Included on its website is a database of technical papers, covering many different engineering topics and methods to teach them. While many papers mention GD&T as a small component in a CAD class, few papers focused on the actual topic itself.

2.4.1 University of Colorado

One ASEE paper, 'Virtual CAD Parts to Enhance Learning of Geometric Dimensioning and Tolerancing' written by Carlson and Trabert, establishes a method for learning GD&T through virtual measurements using CAD software [20]. At the University of Colorado at Boulder, GD&T was originally taught through a series of PowerPoint slides and lectures. After surveying the students this method was rated a 2.7 on a scale of 5 by students because they 'disliked the dry, boring topics delivered in a non-interactive format' . To address this problem, four modules focusing on the GD&T classes of form, orientation, runout, and position were developed. The students completed these modules in a laboratory section. In these modules students open part files that had been purposely modeled with imperfect features in solid modeling software. It was then the student's responsibility to build a ‘perfect’ measurement device to specific criteria, so that the two parts could be combined and the amount of variation measured. This serves to demonstrate the idea of part variation, and how to theoretically check the acceptable limits on part features using CAD software.

To gage the students understanding of GD&T, two questions were added to an end of semester survey. One of the questions asked the students to rate their level of understanding of general dimensioning and tolerancing, and the other question asked them to rate their level of understanding of GD&T. This data was then compared to data from previous semesters, when the previous system for teaching GD&T was used. Students reported an increase in their level of understanding of conventional tolerances, but failed to show any improvement in their understanding of GD&T. The article does not go into enough depth to judge where the gap in understanding conventional tolerancing and geometric tolerancing comes from.

CHAPTER 3

PROBLEM STATEMENT AND GOALS

Geometric dimensioning and tolerancing is a difficult topic to teach and grasp due to its complex rule based system. As a result, students come away with little exposure to the topic in academia, and companies must fill the gap by sending new employees to expensive training seminars, or else risk costly design mistakes. If a new hands-on, visually based GD&T instructional program were developed, students would leave school with a better understanding of GD&T and companies would be able to reduce training costs.

3.1 Thesis Goals

The goal of this thesis research is to develop instructional material that will address the following objectives:

- Understand what a datum is.
- Recognize what purpose a primary datum, secondary datum, and tertiary datum serve and how they are established.
- Realize that manufacturing variation is a reality and that parts are not produced perfectly.
- Identify four of the five main geometric dimensioning tolerance classes.
- Distinguish the different controls allowed through the different tolerance classes.
- Interpret a geometric dimensioning and tolerancing chart.
- Discover how to connect and relate features through the use of geometric tolerances.

- Begin to grasp design intent and how to convey it through an engineering drawing.
- Introduce these fundamentals in an efficient manner, so they can be seamlessly incorporated into an engineering graphics course in one lecture, one group laboratory assignment, and one take home assignment.

CHAPTER 4

EQUIPMENT

The goal of this thesis is to introduce the basics of datums, geometric dimensioning and tolerancing theory, and how to incorporate these fundamentals to reflect design intent. The introduction of datums and GD&T is accomplished through a series of hands-on laboratory exercises which utilize geometric metrology, i.e. part inspection, to visually demonstrate the controls available within GD&T. After students are familiarized with the introductory theory of GD&T, an additional assignment is given which requires the students to incorporate GD&T in order to reflect design criteria given to them in the project problem statement.

These laboratory exercises on geometric dimensioning and tolerancing are developed for *Engineering Graphics and Design (GE 101)*, a course taught at the University of Illinois at Urbana-Champaign (UIUC). By the time that this material is covered, students have a basic working knowledge of two-dimensional engineering drawings with basic dimensions. The sketching laboratory session of GE 101 is held in an area known as the Product Dissection Lab, or PDL, where various pieces of equipment are available. The specific pieces of equipment of interest for this laboratory are an Immersion MicroScribe G2X digitizer or portable coordinate measuring machine (CMM), Autodesk Inventor parametric modeling software , a HighRes plug-in package from ReverseEngineering.com that is used to link Autodesk Inventor and the MicroScribe, a granite surface plate, precision ground aluminum blocks, and a Stratasys Dimension 1200 three-dimensional printer. These pieces of equipment are discussed in more detail below.

The idea for the program is to utilize these pieces of equipment to simulate a manufacturing and quality inspection environment, where students perform geometric metrology, or part measurement, on various manufactured parts. Through the process of inspecting the parts, the fundamentals of GD&T are taught and practiced. The three-dimensional printer is used to simulate the manufacturing equipment, while the granite surface plate and precision ground aluminum blocks simulate an inspection table. The MicroScribe CMM works in conjunction with Autodesk Inventor to take part measurements, after the printed parts are fixtured to the inspection table.

4.1 Immersion MicroScribe G2 Desktop Digitizing Systems

The MicroScribe G2X, seen in Figure 4.1, is used for quickly digitizing three-dimensional objects. In the case of this thesis, the Microscribe G2X will function as a portable Coordinate Measuring Machine to digitize laboratory sample parts.

Figure 4.1: The Immersion MicroScribe G2X.

According to the product manual's functional overview, some of the features the G2X employs, include a unique mechanical linkage system, a counter balanced arm for smooth gliding, a high resolution, and a large workspace [21]. The G2X communicates with a computer using a standard USB port or a RS-232 serial port. The electronics are housed in the unit's digitizing arm, including a series of optical encoders inside each joint, which interface with a microchip in the base of the unit. This microchip then sends a series of joint angles to the host computer, which then uses the forward kinematics of the robot arm to digitize the point that the MicroScribe's probe is in contact with. In order to signal the MicroScribe to record a point, either a foot pedal or a hand held remote is depressed and the computer registers the point into the appropriate software. The MicroScribe G2X has the ability to interface with a variety of software packages, but only Microsoft Excel (see Appendix A and Autodesk Inventor (see Appendix B are considered during this investigation. Table 4.1 describes the technical specifications of the Immersion MicroScribe G2X.

Table 4.1: Technical Specifications for the Immersion MicroScribe G2X

Category	Specification
Position Resolution	0.13 mm (0.005")
Position Accuracy	0.38 mm (0.015")
Reach of Digitizing Arm	1270 mm (50")
Footprint Size	152.4 mm x 152.4 mm (6" x 6")
Interface	RS-232C / USB 1.1
RS-232C Baud Rate	Up to 115 Kbps
Button Options	Foot pedal, desktop unit, hand-held units
Power Requirements	External 115V or 220V power supply
Power Usage	+5V DC, 600 mA max

4.2 HighRes

The HighRes software is a plug-in from ReverseEngineering.com, which allows the MicroScribe G2X to interface with Autodesk Inventor, as well as other CAD packages [22]. With this plug-in, the digitizer can perform the following tasks:

- Set 3 point axis with Digitizer
- Digitize 2D Sketches including geometry such as points, lines, closed lines, splines, closed splines, arc, rectangles
- Connect with Com ports and USB ports
- Eliminate data translation issues by ensuring all digitized data is in true native Inventor entities

- Create freeform 3D including 3D workpoints, lines, closed lines
- Create circle from 3 points
- Measure point to point

The minimum system requirements for running the HighRes plug-in include the following:

- 20 GB Hard Drive
- Microsoft Windows XP
- Pentium 4 Processor
- 1 GB Ram
- HighRes Compatible Digitizer
- 128 MB High Resolution Grahpics Card
- OpenGL graphics

4.3 Granite Surface Plate

A black granite surface inspection plate was donated to the PDL and is used as the base surface for the CMM inspection table.

4.3.1 History of Granite Surface Plates

Iron was the standard surface plate used during inspection processes prior to the Second World War. However, due to the lack of iron, a new material had to be found. According to the L.S. Starrett Company (founded in 1880 and manufacturer of more than 5,000 variations of precision tools for markets worldwide) Wallace Herman needed a flat inspection plate for his monument and metal working in Dayton, Ohio during this time period. He hypothesized that granite would be an acceptable solution, and he fashioned a surface plate from a piece of black granite stone. The idea was successful and the use of the granite plate has evolved and is still in use today [23].

4.3.2 Granite Surface Plate Properties

The important properties for a surface plate include the high material stiffness to reduce bending, high hardness to reduce damage, high density to ensure a fine surface finish, and good wear resistance to minimize damage from use. According to the Starrett Company, granite is better than steel or cast iron in all these categories and additional advantages include, "no rust or corrosion, almost immune to warping, no compensating hump when nicked, longer wear life, smoother action, greater precision, virtually non-magnetic, low coefficient of thermal expansion, and low maintenance cost" [24].

Typically two types of granite are used when manufacturing surface plates, black granite and pink granite. Diabese, sometimes called black granite, is not technically granite because it lacks quartz and is denser than granite. However, it is stiffer than granites and has a modulus of elasticity 9.5×10^6 psi. Pink granite only has a modulus of elasticity of 3.8×10^6psi, but contains 32% quartz which makes the material five times more wear resistant than black granite.

4.3.3 Granite Surface Plate Setup and Maintenance

A surface plate is supported at three points during manufacturing, and the laboratory should use these same three support points during the setup and use of the inspection plate. If a different set of three points or more than three points are used to support the granite, the plate will deflect to conform to the new supporting points and surface errors will be introduced into the system. If a plate is properly supported, leveling is not necessary unless specifically called for. The manufacturer will typically specify the location of the three critical points and often times the granite pieces are sold with support stands that match these critical points.

Once a surface plate is setup, it does not require much maintenance or care. It is best to take care to keep the surface clear of dust, dirt, grease, grime, and other foreign materials to allow for accurate tool measurements. The surface should be cleaned occasionally using an appropriate surface cleaner, and if not in use it is best to cover the surface.

4.4 Precision Ground Aluminum Side Rails

Since dedicated computer aided inspection software is not used during the laboratory, two side rails are utilized to provide the secondary and tertiary datums for fixturing a part. The initial solution was to contact a few machine shops to quote precisely machined steel or aluminum blocks. However, due to the capabilities of the shops and the tight tolerance requirements, the finished product from these shops would have been very expensive. Alternative options were researched, and aluminum precision ground CNC blanks are now used as the side rails. These blanks are found on the McMaster-Carr website. According to the blanks data sheets, the blocks are 101.6 mm (4 in) tall by 304.8 mm (12 in) wide and are 12.7 mm (0.5 in) thick [25]. These dimensions are accurate up to 0.0508 mm (0.002 in). The side rails are then arranged using precision ground 1-2-3 blocks to ensure a 90 ° mating angle. These blocks are then clamped to the granite surface plate using standard bar clamps purchased at Home Depot. A picture of the setup can be seen in Figure 4.2.

Figure 4.2: Using the aluminum side rails and granite surface plate to fixture a part.

4.5 Stratsys Dimension SST 1200 Three-Dimensional Printer

The three-dimensional printer used in the PDL is the Stratasys Dimension SST 1200, which can be seen in Figure 4.3.

Figure 4.3: The Stratasys Dimension SST 1200.

The Wholers Report 2005 was referenced for the specifics of the three-dimensional printing process. Terry Wholers, the author of the report, is the president of Wohlers Associates, Inc., which provides technical and strategic consulting on the new developments and trends in rapid product development and additive manufacturing [26].

The Stratasys Dimension 1200 operates utilizing fused deposition modeling, where acrylonitrile butadiene styrene (ABS) plastic is used as the model material. The process begins by taking a standard computer aided design (CAD) file and converting it from its native format to a stereolithography (STL) format. In the PDL, Autodesk Inventor is used as the CAD software package to complete this task. "The STL file approximates the shape of a solid model using small triangles called facets. The smaller the facet size, the better the surface approximation, but at the expense of file size and processing speed" [26]. After the

STL file is created by the modeling software the file is then imported into a rapid prototyping software. In the case of the PDL, the Catalyst EX software from Stratasys is used. The rapid prototyping software then uses a special algorithm to horizontally slice the STL file to the desired cross section thickness. In the case of the Stratasys Dimension 1200, the layer thickness can either be 0.330 mm (0.013 in) or 0.254 mm (0.010 in). The software then sends the data for each layer to the Dimension 1200, which uses a fused deposition modeling (FDM) process to print the parts. The printer deposits multiple filament materials through two heated extrusion tips. The previously mentioned ABS plastic serves as the model material, and a second, proprietary, water-soluble material is used to support the ABS as it cools.

4.5.1 Stratasys Dimension 1200 Accuracy

To determine the accuracy of the Stratasys Dimension 1200, a study on three-dimensional printer accuracy was used [27]. The 2005 study compared the accuracy of three leading rapid-prototyping printers: the Stratasys Dimension 1200 (FDM technology) [28], the 3D Systems InVision SR [29] (Multi-Jet Modeling and thermal material application with UV-curing technology), and the Z Corporation Zprinter 310 [30] (Massachusetts Institute of Technology's patented Three-Dimensional Printing technology). During the study, four parts each were created in a CAD program and then used as the benchmark objects for the test. The parts were labeled battery bottom, battery top, fixture base, and fixture cap and each part was roughly 2.5 in. x 2 in. x 1 in. Each printer was required to print the four parts and then a combination of laser scanning and computer aided inspection was used to analyze the accuracy of the produced parts. The inspection procedure generated 200,000 data points per part to create a thorough profile of the parts created, so a detailed and complete analysis of the results could be performed. A detailed list of the results for the dimensional accuracy tests on the Dimension 1200 can be seen in Table 4.2

Table 4.2: Dimension 1200 Dimensional Variation Results

Measure	Battery Bottom	Battery Top	Fixture Base	Fixture Cap
Mean (in.)	-0.0009	0.0001	0.0000	0.0009
Std. Dev.	0.0030	0.0034	0.0031	0.0027
Max. Error (in.)	0.0147	0.0198	0.0204	0.0152
Min. Error (in.)	-0.0151	-0.0167	-0.0297	-0.0098
Points Captured	213132	245925	260273	264273

After the analysis, it was discovered that even though the Stratasys Dimension 1200 is a less costly option than the Z Corporation Zprinter 310 or the 3D Systems InVision SR, it outperformed both printers when it came to comparing the dimensional accuracy of the parts created. When measured, the parts produced Stratasys Dimension 1200 had a small standard deviation for the dimensional variation. The maximum and minimum data points taken from the sample parts were centered on the nominal value from the CAD file, which when combined with the low standard deviations demonstrated the repeatability and quality of the Stratasys Dimension 1200 . This led to the conclusion that for parts similar in size to those used in the study, a tolerance of ±0.003 in. (±0.08 mm.) was reasonable, and for parts deviating from this size a tolerance of ±0.01 in (±0.25 mm.) could be expected.

4.6 Autodesk Inventor

Autodesk Inventor is a three-dimensional parametric modeler, which according to Autodesk provides a comprehensive and flexible set of software for 3D mechanical design. The ability of Autodesk Inventor to create both two dimensional and three-dimensional sketches is very important for the use of the MicroScribe G2X digitizer. Due to this feature, the data points from the digitizer can be seamlessly read into Autodesk Inventor with the HighRes plug-in and students can see the points appear in real time. Requirements to run Autodesk Inventor version 2010 on Microsoft Windows operating system are listed below [31]:

- Microsoft Windows 7 (32-bit or 64-bit) Home Premium, Professional, Ultimate/ Enterprise, or Microsoft Windows Vista (SP2) (32-bit or 64-bit) Home Basic, Home Premium, Business, Enterprise, or Ultimate, or Microsoft Windows XP Professional (SP3) or Professional x64 Edition (SP2)
- Intel Pentium 4, 2 GHz or faster, Intel Xeon, Intel Core, AMD Athlon 64, or AMD Opteron, or later
- 2 GB RAM or more
- Microsoft Direct3D 10 or Direct3D 9 capable graphics card
- DVD-ROM drive
- Microsoft Mouse-compliant pointing device
- 1280 x 1024 or higher screen resolution
- Internet connection for web downloads and Subscription Aware access
- Adobe Flash Player 10
- Microsoft Internet Explorer 6.x through 8
- Microsoft Excel 2003 through 2007 for iComponents, thread customization, and spreadsheet - driven designs

CHAPTER 5

RESULTS

The final product of this work is a hands-on, visually based instructional system focused on illustrating the basic principles of GD&T. The specific deliverables of the program include two lab units, a complete laboratory setup for inspection, instruction manuals for interfacing the portable CMM with Autodesk Inventor, and Inventor part files with and without manufacturing variation. The first lab unit includes: 1) a classroom lecture portion on datums, geometric tolerances, tolerance zones, and variation location, 2) an exercise on datums, tolerance zones, and variation, and 3) an exercise calling for the inspection of two rapid prototype parts with manufacturing variation built in. The second lab unit includes a classroom lecture portion on design intent and an exercise focused on applying these engineering tools in real world scenarios. The central objective of the program is to engage students so that they become interested, interact, ask questions, actively participate and ultimately learn the material. The interactive nature of the program is a divergence from the traditional strategy of teaching geometric dimensioning and tolerancing built on its rule based system, which both professors and students find dry and difficult to teach and grasp. Furthermore students are not only taught the fundamentals of these tools, but also how to properly utilize them during the design process. An assumption made during the development of this work, is that students have some familiarity with two dimensional engineering drawings, basic dimensions, and dimensional tolerances.

5.1 Tolerance Visualization through Data Point Digitizing

The aim of this program is to improve upon the standard practice of geometric dimensioning and tolerancing instruction by incorporating the use of hands on activities. In order to accomplish this task, the Immersion Microscribe is used to capture variation present in physical part models, so that students are able to visualize what manufacturing variation is, and how different tolerances accomplish the control of this variation through the use of tolerance zones and boundaries. The MicroScribe G2X has many ways to read data into a computer, but two methods of interest are: 1) reading data into Microsoft Excel and 2) reading data directly into Autodesk Inventor. These two programs were chosen since they are available in the PDL, but using similar software packages would have the same effect.

5.1.1 Approach 1: Reading Data into Microsoft Excel

The first technique, reading data into Microsoft Excel, is a mathematically intensive process, because the data point relationships are calculated instead of visually observed and measured. This procedure takes the X, Y, and Z coordinates of an individual point recorded by the CMM, and reads each coordinate of the point into its own column. For example, if cell A1 is selected during the measurement process, the X coordinate of the data point is placed in cell A1, the Y coordinate is placed in B1, and the Z coordinate is placed in C1. Once this is complete, the CMM signals that the active cell should move to cell A2 and the process repeats. This procedure continues until the user has completed taking all the desired points. Detailed directions can be found in Appendix A.

Once the student has a collection of points, the inspection process begins. The set of points represents a sampling of the feature being inspected. One can imagine that it can be difficult to take these points, visualize them in three dimensions, and then form a mental picture of the feature being inspected. This can be difficult for many individuals, regardless of their background or education, but it proves especially problematic in this case, since the students for which this lab is intended are mostly first college students with little engineering experience. These students are just beginning to develop the ability to think graphically about three dimensional objects, and they have limited familiarity with three dimensional plotting and the mathematics which accompany these plots. This leads to the scenario of the instructor having to teach both three dimensional plotting and geometric tolerancing, while the student struggles to learn multiple topics during the same session. Both the instructor and the students are frustrated with the exercise, and the ultimate goal of engaging and teaching the students is not realized.

5.1.2 Approach 2: Reading Data into Autodesk Inventor

Fortunately, the MicroScribe G2X also has the ability to interface with parametric solid modeling software packages. In the case of Autodesk Inventor and the HighRes plugin package, the CMM can interface with a standard Inventor part file (.ipt file extension). This part file can contain previously created geometry or it can be blank, but the default coordinate system of the part file is always used as the reference for the X,Y, and Z components of the points taken by the CMM. Additionally, it is important to match the unit settings of the CMM to the units of the part file, which in the case of this study are millimeters. During the calibration of the G2X, discussed in more detail in Appendix B, the X axis, Y axis, Z axis, XY plane, YZ plane, and XZ plane of the measurement table are made to coincide with to the same default features in the Autodesk Inventor part file. When these work features in Inventor are made visible, the relationship between the measurement table planes and the default planes in Autodesk Inventor becomes clear to the student.

During the checking of a physical part, the student will again take a sampling of points on the feature of interest. This time however, as the student takes inspection points, they appear in the Autodesk Inventor graphics window. In utilizing the capabilities of the solid modeling software, the challenge of mentally visualizing the inspected surface is removed. This allows students to limit their focus to the variation in the part and how it appears in reality.

The other benefit of utilizing Autodesk Inventor is that the perfect part can be modeled prior to the inspection process. Since geometric tolerancing is heavily based on theoretically correct features, this allows students to use perfect part features in Autodesk Inventor as a reference for the inspection points taken by the CMM. Students can then pan, zoom, and rotate to see the relationship between the inspection points and the perfect features. This connection between the perfect feature and the inspection points naturally introduces the idea of a tolerance zone. This concept of a zone is then expanded upon, as the students are required to model the actual tolerance zones, and to then take measurements in order to see how in/out of tolerance the inspection points are.

While this method assumes that students are familiar with Autodesk Inventor (or another solid modeling software package), this is not a problem since solid modeling in Autodesk Inventor is an integral component of the GE 101 course at the University of Illinois at Urbana-Champaign, which this curriculum is designed for. Assuming that this obstacle is a not an issue, the benefits of utilizing solid modeling software outweigh those of using the coordinate point method in Microsoft Excel because the potential problems of mental visualization and mathematical theory are removed.

5.2 Lab Unit 1: Datums, Variation, and Geometric Tolerancing

The first section of this work focuses on introducing the student to the concept of datums, variation, and geometric tolerancing. The material is broken down into an informational section designed to be presented to students either by a teaching assistant in a laboratory session or an instructor in a lecture environment. By including a classroom section, students are first exposed to the concepts, and are able to ask questions. This informational session is then followed by an independent lab study session that can be completed individually or in groups. This individual period gives the students the opportunity to actively complete a guided tutorial covering the material presented in the lecture portion.

5.2.1 Lab Unit 1 Lecture -Datums, Variation, and Geometric Tolerancing

The information for the first lecture portion is based on the American Society of Mechanical Engineers (ASME) Y14.5M-1994 standard for Dimensioning and Tolerancing [2]. While no new concepts in GD&T are produced, the way in which it is presented differs significantly from the rule-based methods found in text books such as Engineering Graphics Principles with Geometric Dimensioning and Tolerancing by Raisor [32], Mastering CAD/CAM by Zeid [33], and Engineering Design Graphics: Sketching, Modeling, and Visualization by Leake and Borgerson [34] and industry training manuals such as Geometric Dimensioning and Tolerancing Workbook by Al Neumann [4].

Introduction to Datums

The lecture material begins with an introduction to datums. To present the topic, a simple rectangular block is created and three surfaces are chosen as datums. These datum surfaces are designed orthonormal to one another, so an easy comparison to a Cartesian coordinate system can be made. By doing this, students are able to grasp the concept of a coordinate system on the part, as well as to visualize how the coordinate system is defined. The block can be seen in Figure 5.1.

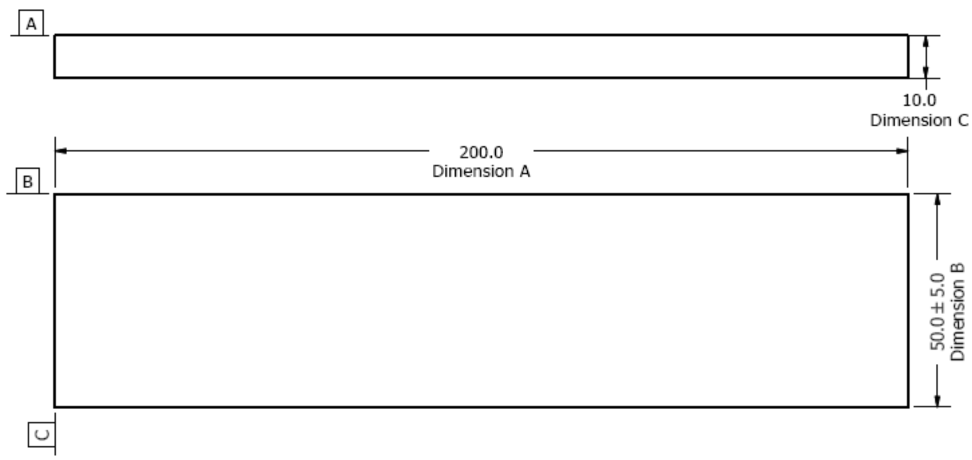


Figure 5.1: The rectangular block with datums.

Datum Reference Frame, Degrees of Freedom, and the Inspection Table

The next segment of the lecture focuses on datum usage. To introduce the concept of a datum reference frame, the rectangular block is placed on the inspection table in different configurations. The order in which the part datums contact the simulated datums of the inspection table, help students to grasp the idea of primary, secondary, and tertiary datums. Through this process, the relationships between the degree of freedom removal, datum reference frame, and inspection baselines are established. An illustration explaining this concept can be seen in Figure 5.2.

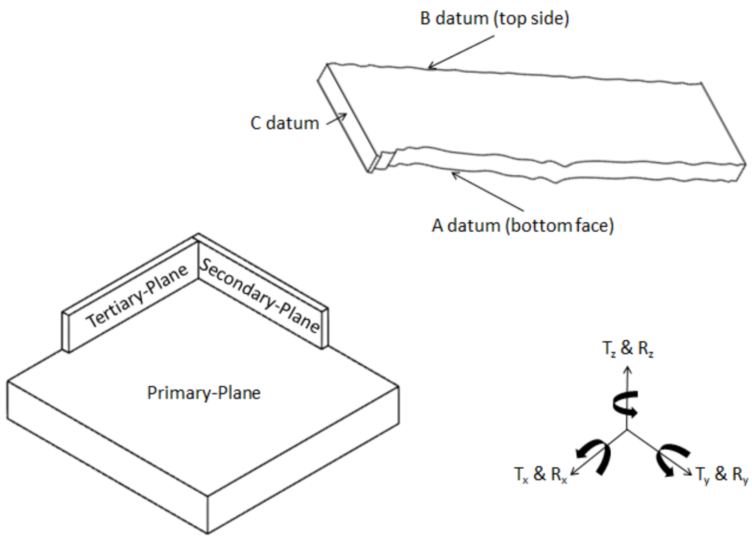


Figure 5.2: Illustration of inspection table, degrees of freedom, and datums.

Geometric Tolerances and Inspection Procedures

Of the five geometric tolerance categories of form, profile, orientation, location and runout, the laboratory focuses only on form, profile, orientation and location. Within each respective class the flatness tolerance (form), profile of a surface tolerance (profile), perpendicularity tolerance (orientation), and true position tolerance (location) are explained, analyzed, and inspected in detail. Since many of the basic concepts hold for tolerances within a class, a discussion of one tolerance type within each category is sufficient to gain insight into the remaining tolerance types. In addition to the explanation of these four geometric tolerances, the inspection procedures for each tolerance type are also covered.

Profile of a Surface Tolerance

The first tolerance discussed is the profile tolerance, and it is applied to one of the surfaces on the rectangular block. The profile of a surface tolerance is given a value of 10 mm, and is applied with respect to datum ***-A-***, datum ***-B-***, and datum ***-C-***. The block is then fixtured to the inspection table using the appropriate datums specified in the datum reference frame, as seen in Figure 5.3.

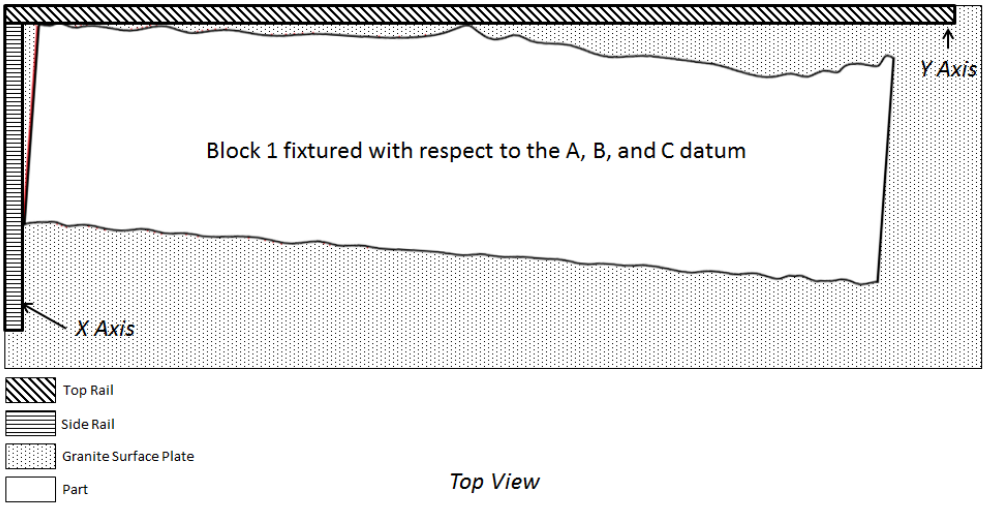


Figure 5.3: Block 1 fully fixture to the measurement table.

An Inventor part file containing the perfect block is opened in Autodesk Inventor, which is modeled in the same configuration as that of the imperfect block on the inspection table. The CMM is then interfaced with the solid modeler. Inspection points are taken with the CMM and they appear near the perfect surface modeled in Inventor. After the inspection points are complete, the plane feature in Inventor is used to model the boundary zones of the tolerance around the perfect surface. At this point, it is now evident whether the inspection points fall within the boundary limits of the tolerance. The measurement function in Autodesk Inventor can also be utilized to measure the inspection points against the tolerance limits. This can be seen in Figure 5.4.

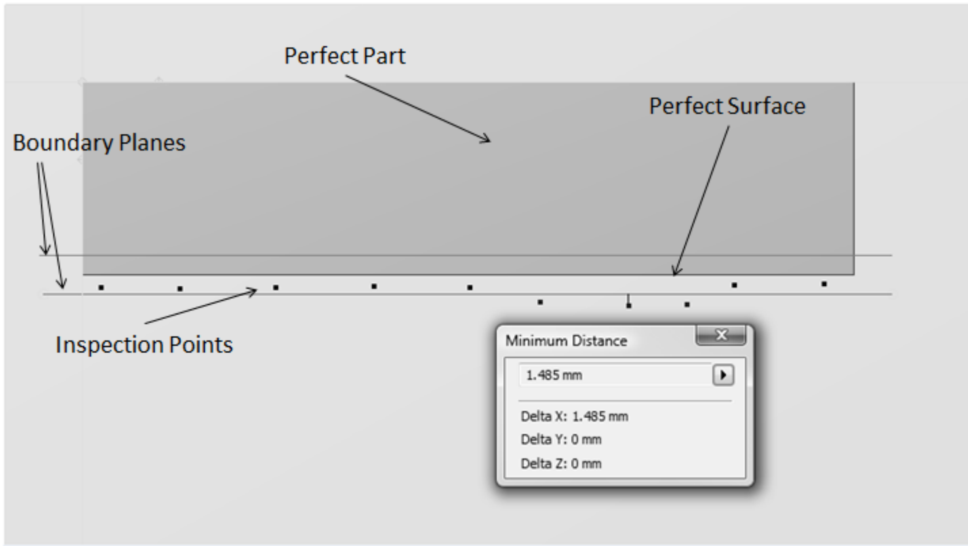


Figure 5.4: Profile inspection results, as seen from above. The most extreme point is 1.485 mm out of tolerance.

Perpendicularity Tolerance

Much like the profile of a surface tolerance, the perpendicularity tolerance process begins with the fixturing of the imperfect block to the table, the synchronizing of the CMM with Autodesk Inventor, and the recording inspection points. However, since perpendicularity only controls orientation, and not location, a separate post-processing inspection procedure is required.

The post-processing procedure is designed to measure the variation between a set of inspection points and then compare this value against the perpendicularity tolerance value. To do this, a measurement plane needs to be chosen as a baseline reference. In this case of the profile tolerance the perfect surface is used as the measurement plane, because the block has all six degrees of freedom locked by the datum reference frame. So the perfect surface in the solid model and the physical surface on the inspection table coincide. However, in the case of a perpendicularity tolerance, all six degrees of freedom are often times not controlled by the datum reference frame. With this freedom, the block has the ability to move on the inspection table and the perfect surface in the solid model will not coincide with the physical surface on the inspection table. Therefore, a baseline plane must be created from the inspection point data and not from a perfect surface in Autodesk Inventor.

The process of creating a baseline plane begins by creating a two dimensional sketch on the datum surface, which is perpendicular to the toleranced feature. On this sketch, the first inspection point and the last inspection point are projected onto the sketch plane. Through these two projected points, an approximate line of best fit is created for the remaining inspection points, as seen in Figure 5.5.

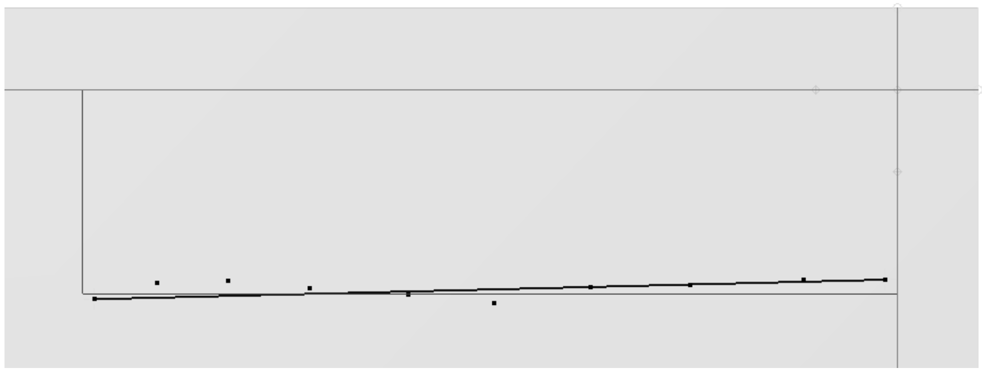


Figure 5.5: Best fit line created between the first and last projected points.

The sketch is then exited, and the best fit line and the datum reference plane are selected, so that a plane is created which is perpendicular to the datum reference plane and oriented along the best fit line, as seen in Figure 5.6. This plane approximates the location and orientation of the physical surface on the inspection table.

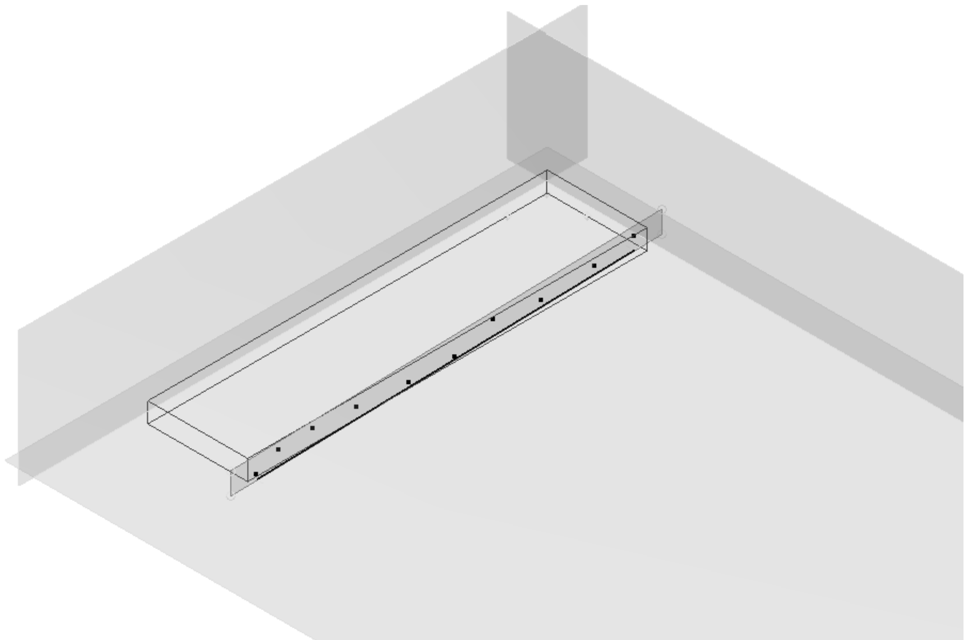


Figure 5.6: Creating the perfect plane from the best fit line and reference plane.

Aside: Linear Regression

Alternatively, all the inspection points could be projected onto the perpendicularity datum reference, transferred into Microsoft Excel, and a linear regression could be performed on the data points. The best fit line could then be transferred to Autodesk Inventor and subsequently modeled. However, this technique is more math and time intensive and has the potential of clouding the learning process, and therefore was not chosen as the most suitable technique.

Aside: Autodesk Inventor Measurement Tool

Inspection points may lie on either side of the simulated best fit surface, as seen in Figure 5.6. At this point, the Autodesk Inventor measurement tool can measure the variance of the inspection points with respect to this best fit plane. However, this tool always returns a positive scalar distance and direction is not considered. Due to this fact, these scalar values alone cannot be used to calculate the difference between extreme points. In addition, with small variations the resolution in Autodesk Inventor is not great enough to distinguish which side of the plane the points occur. So to eliminate this problem, the solution is to create an offset plane to eliminate the need for a direction vector.

Creating an Offset Plane to Eliminate Direction Scalar Problem

To remedy this problem, a second inspection plane is created, parallel to the first inspection plane. This plane is offset by some value greater than the maximum variation seen in the feature, so that all points will occur on the same side of the plane. For this laboratory a value of 20 mm is selected for the offset and is sufficient. This eliminates the need for a direction of the vector. Microsoft Excel is used to calculate the maximum variation between the two most extreme points and this value can then be compared against the perpendicularity tolerance value. This measurement process can be seen in Figure 5.7.

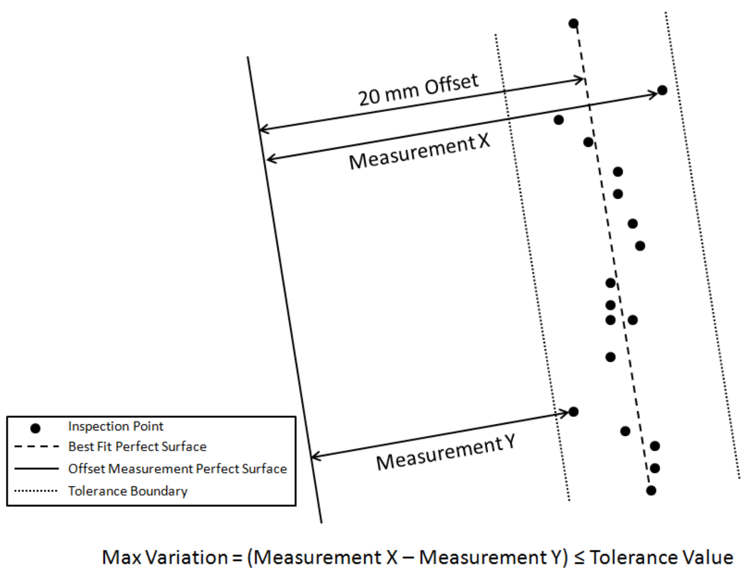


Figure 5.7: Schematic of the perpendicularity inspection process with an offset plane.

Flatness Tolerance

The inspection process for flatness again begins by fixturing the imperfect block to the inspection table, synchronizing the CMM with Autodesk Inventor, and taking inspection points. The form tolerance class is different from the other tolerance classes, because no datum reference feature is needed for control. So once the data points have been taken, any combination of three points can be used to form the reference plane for inspection. To capture variation across the entire surface, three evenly spaced points are used out of the point cloud to form a perfect datum plane, as seen in Figure 5.8. This avoids capturing variation on just one corner or section of the surface. This is another approximate approach adopted for this laboratory, but often times in industry different algorithms or techniques are used for a more precise process.

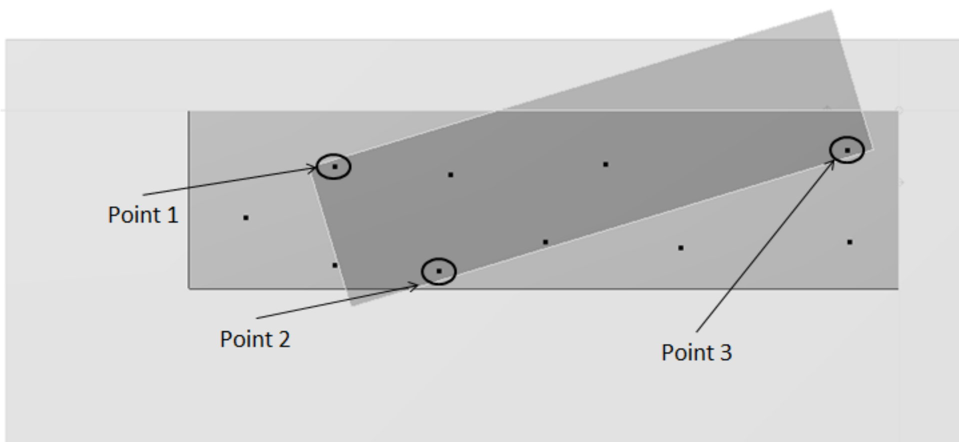


Figure 5.8: Top View of CMM inspection points and perfect plane. The plane takes no particular orientation with respect to the perfect part. This is because the flatness tolerance controls no degrees of freedom.

Once the datum reference plane is established, another inspection plane is created parallel to the first inspection plane with an offset that is larger than the maximum variation in the surface. This is similar to the technique used during the perpendicularity inspection process, so that confusion over the direction of the point to plane measure is avoided. The offset for this plane is again suggested to be 20 mm. All the inspection points are then measured against this offset plane and their values are entered into an Excel spread sheet. As with the perpendicularity tolerance, the excel spreadsheet calculates the maximum variation between points and compares this value to the flatness tolerance value.

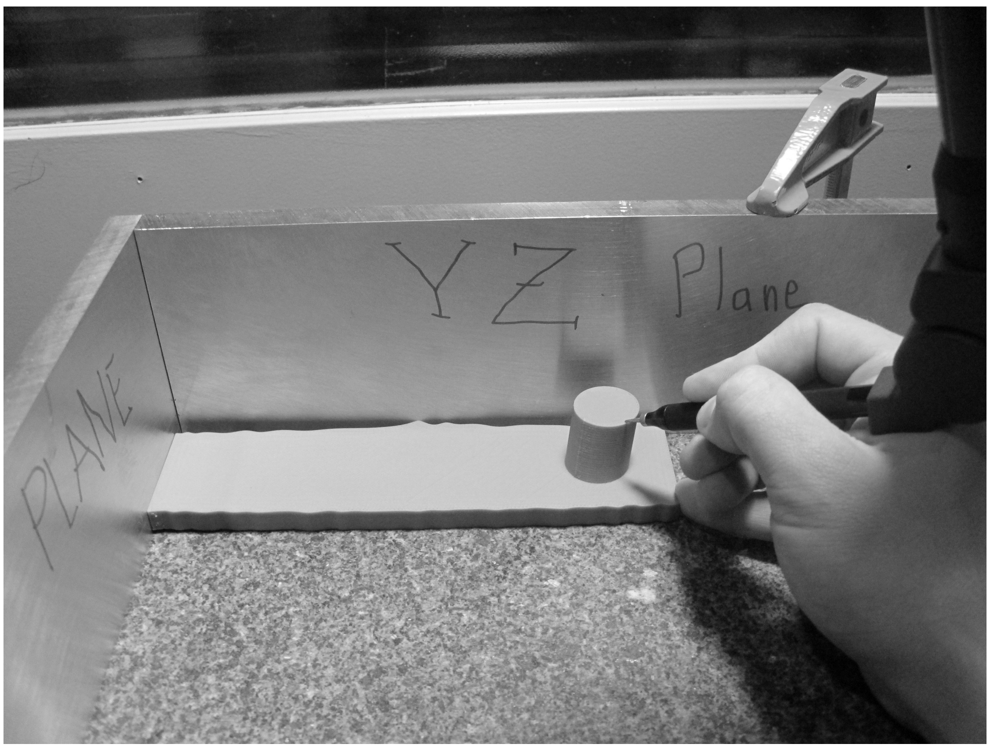
True Position Tolerance

In most applications the true position tolerance is typically applied with a material condition modifier. There are three material condition modifiers: 1) maximum, 2) minimum, and 3) regardless of feature size (RFS). The maximum and minimum material condition modifiers combine the actual size of the feature and the tolerance value to further refine or expand the allowable boundaries. However, to avoid additional confusion, in this laboratory all true position tolerances are added RFS, so the bounds are not refined. This is done so that only the general rules of a true position tolerance are considered, while additional rules are avoided. If interested, the ASME Y14.5-1994 geometric tolerance standard can be consulted to expand on the basic notion of a true position tolerance [2]. Additionally, in the laboratory exercises the only feature considered for a true position tolerance is a cylinder or hole.

In order to inspect the true position tolerance of a cylindrical feature (i.e. an extruded cylinder or a hole), the two-dimensional three point circle function is used in the HighRes plug-in package. This feature allows the user to select three points on a cylinder or hole, and the software forms a circle from these three data points.

To begin the process, the imperfect block is fixtured to the inspection table according to the datum reference frame, the perfect block is opened in Autodesk Inventor, and the CMM is synchronized with Autodesk Inventor. The three point circle command under the HighRes plug-in tab is then used to capture the two defining circles on the cylinder or hole of interest. Three data points are taken around the first circle at the base of the cylindrical feature, as seen in Figure 5.9(a), and three points are taken on the second circle at the top of the cylindrical feature, as seen in Figure 5.9(b).

(a) Taking an inspection point on the base circle of the cylinder.



(b) Taking an inspection point on the top circle of the cylinder.

Figure 5.9: Taking inspection points on the base circle and top circle of the cylinder.

After the data is taken, two circles appear in the Autodesk Inventor window that simulates the size and location of the base of the cylinder and the top of the cylinder, as in Figure 5.10.

Figure 5.10: The base circle and top circle.

The two center points of these circles represent the start point and end point for the actual axis of the cylindrical feature. This axis must fit within a perfect cylinder, which has a diameter of the tolerance zone and is located by basic dimensions on the engineering print. This can be seen in Figure 5.11.

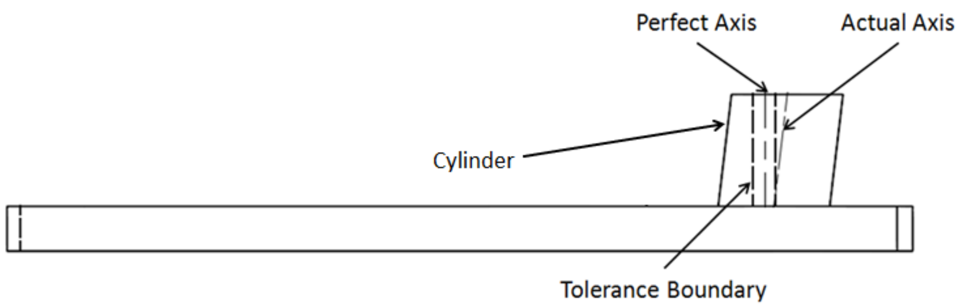


Figure 5.11: The boundary zone for the true position tolerance.

To create this boundary zone, the sketch tool is used to create a circle on the same plane as the inspection circles. The size and location of this boundary zone will be called out by the engineering drawing of the part under inspection. To verify that the cylindrical feature meets the tolerance requirement, the center points of the inspection circles (i.e. the axis of the cylinder) must fall within the diameter of the boundary circle (i.e. the cylindrical tolerance zone). The top view of the inspection circles and boundary circle can be seen in Figure 5.12.

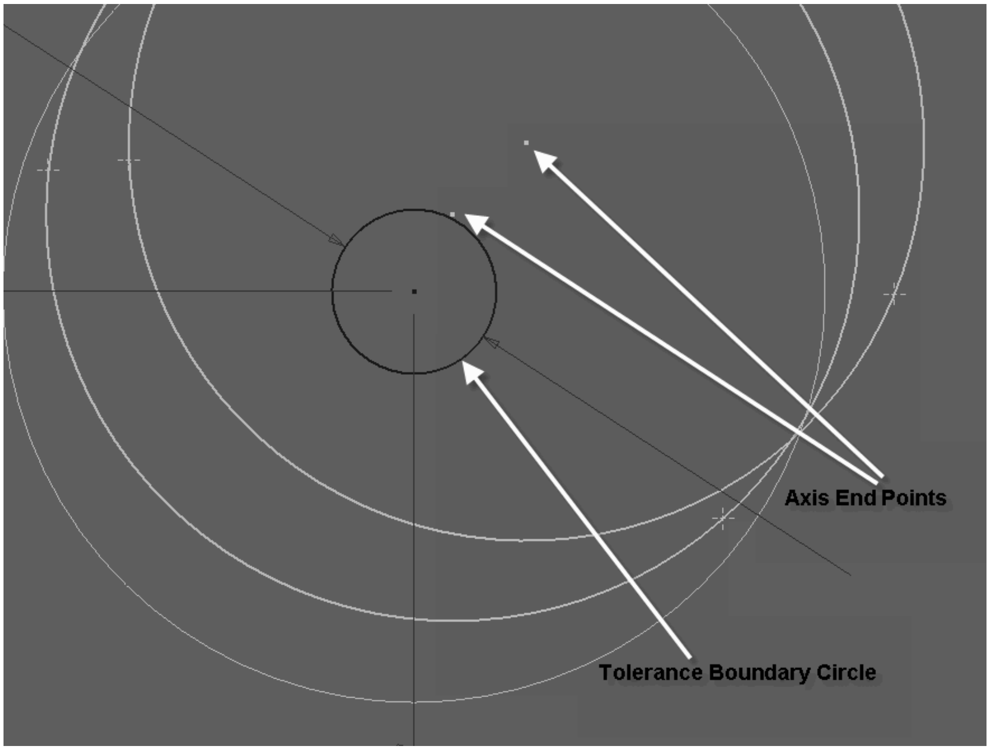


Figure 5.12: Examining the inspection points and the tolerance boundary.

5.2.2 Lab Unit 1 Assignment A-Datums and Manufacturing Variation

A series of engineering drawings of a rectangular block are provided to students to introduce, explain, and demonstrate the functionality of datums. The first engineering drawing, seen in Figure 5.13, only contains a one dimensional tolerance on the height dimension, 50 mm, with no datums incorporated. The tolerance on the height dimension of the block is large enough, ±5 mm, so that the allowable variation is evident through visual examination of the part.

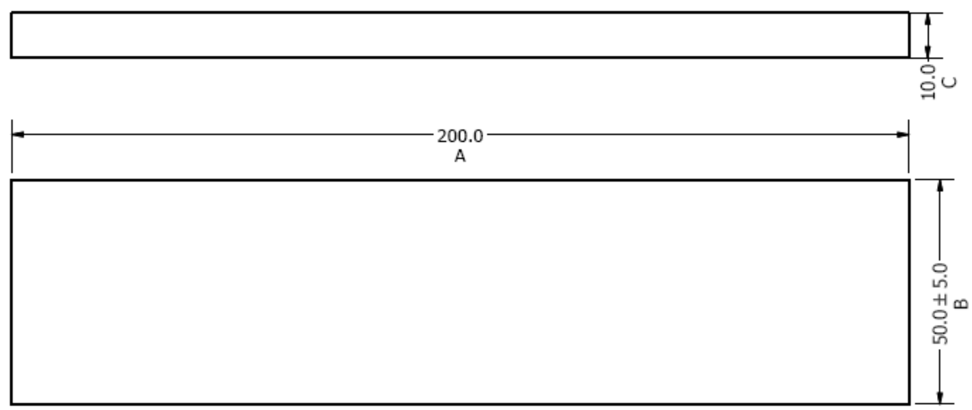


Figure 5.13: A basic engineering drawing of a rectangular block with a dimensional tolerance on the height.

The idea behind magnifying and exaggerating the defects in the part is to allow students to easily grasp the concept of manufacturing variation. Accompanying the imperfect block, is a design criterion stating that when the block is resting on a table with the 50 mm height dimension in the vertical direction, the block should be no taller than 55 mm and no shorter than 45 mm. An example of this criterion can be seen in Figure 5.14.

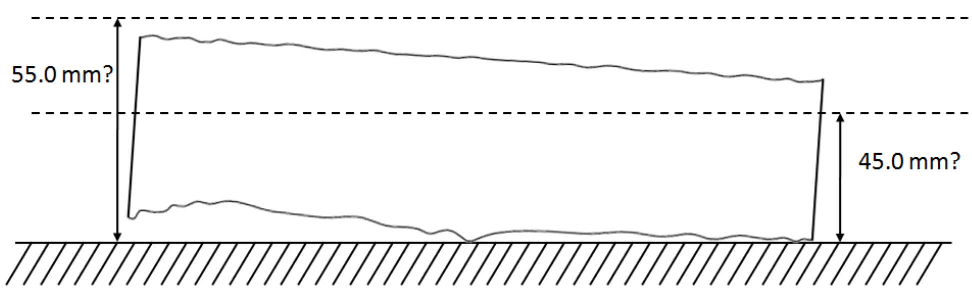


Figure 5.14: Design criterion placed on the rectangular block.

The worksheet is loosely guided, and there are various exercises involving free measurement of the block with calipers and fixed measurements of the block with the CMM. This provides an opportunity to practice fixturing techniques and also demonstrates how measurements change with different fixturing. Finally, different versions of the block are produced with the variation located in different areas, which demonstrates the need for greater control than that provided through dimensional tolerances.

5.2.3 Lab Unit 1 Assignment B and C-Tolerance Inspection

Two additional engineering drawings and imperfect blocks are used for assignments B and C. The engineering drawings can be seen in Figure 5.15 and Figure 5.16.

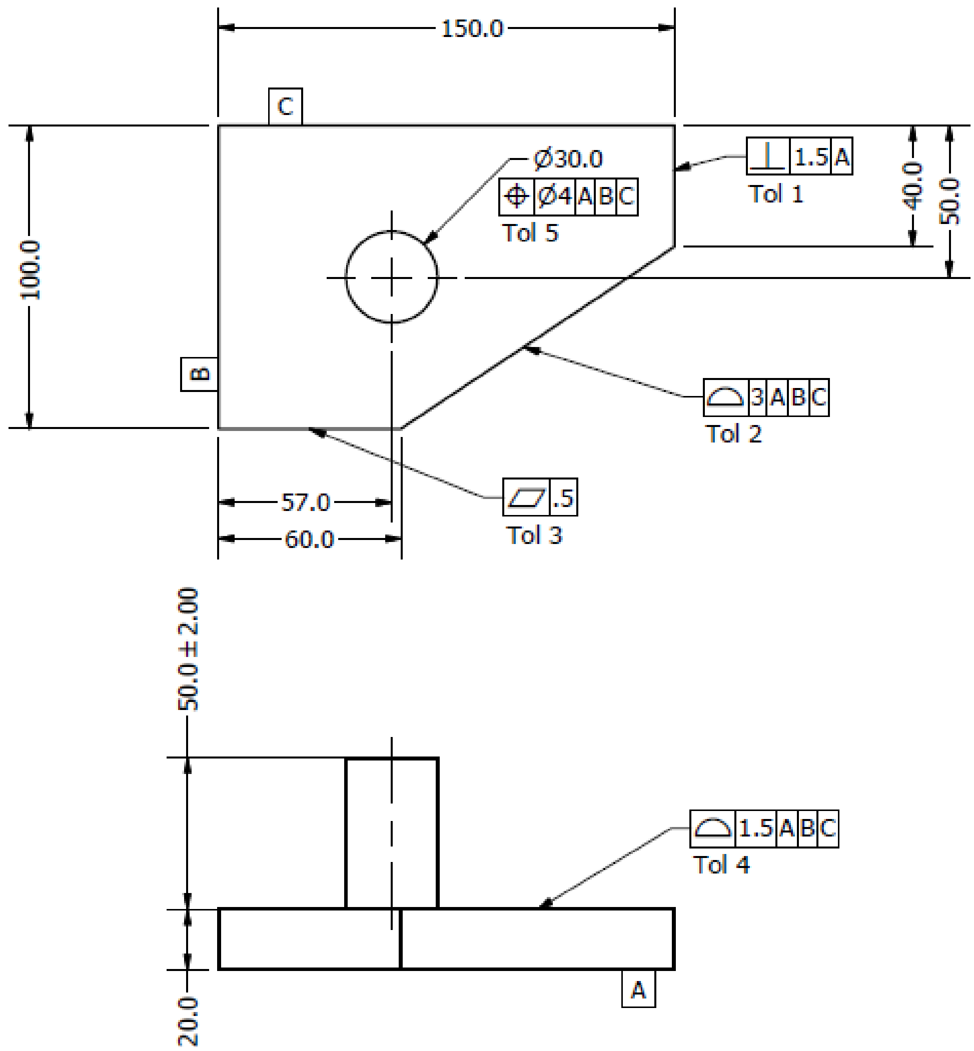


Figure 5.15: Engineering drawing of Part 2.

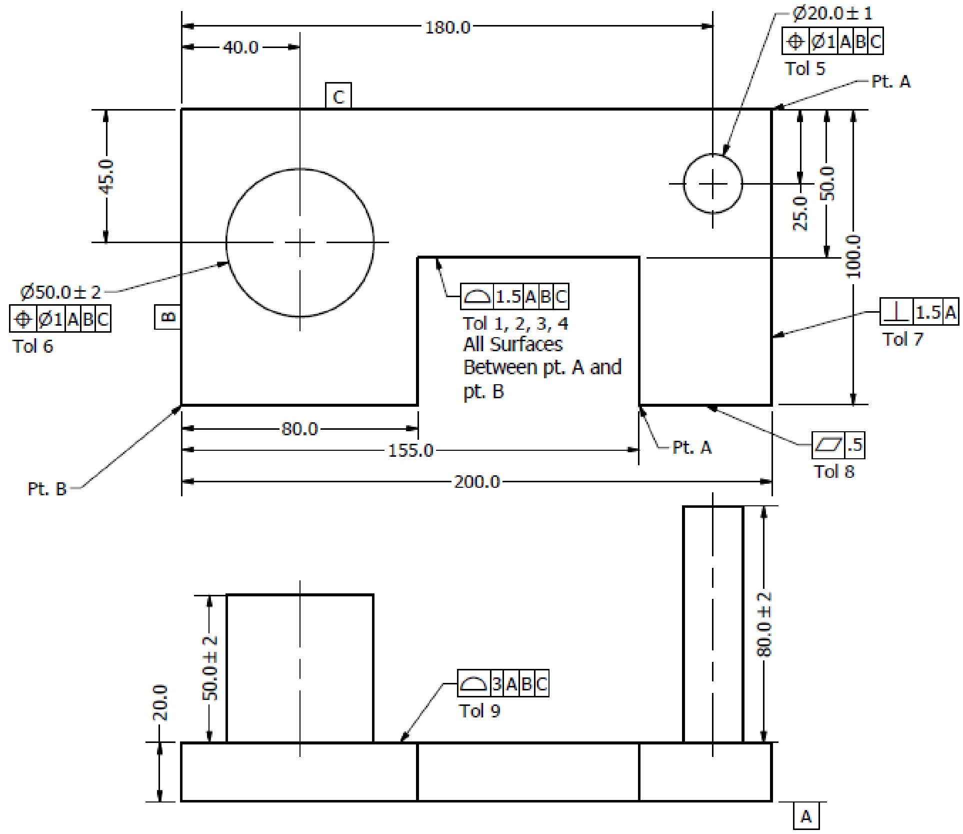


Figure 5.16: Engineering drawing of Part 3.

The intent of lab assignments B and C is to provide an opportunity to physically explore the meaning of the various geometric tolerances indicated on the drawings. Each tolerance has an accompanying set of questions about the tolerance zone shape, perfect feature, datum reference frame, whether the feature passes inspection, and the amount manufacturing of variation the feature contains. To answer the latter two questions the part must be fixtured appropriately, data must be taken with the CMM, and tolerance boundaries must be constructed in Autodesk Inventor.

5.3 Lab Unit 2: Reflecting Design Intent through the use of Geometric Tolerancing

Explaining design intent through a set of rules is very difficult if not impossible to do, so the concept is presented through an example. During the explanation of this example, different design criteria are presented and then translated into a datum scheme and a fully toleranced drawing. A second example is then provided as an independent study, where new design criteria must be interpreted into a datum scheme and a set of tolerances.

5.3.1 Lab Unit 2 Lecture-Engineering Drawing for Machine Blank

A simple machine is modeled in Autodesk Inventor and the block from Lab Unit 1 now becomes a blank that is to be inserted into a simple machine, seen in Figure 5.17.

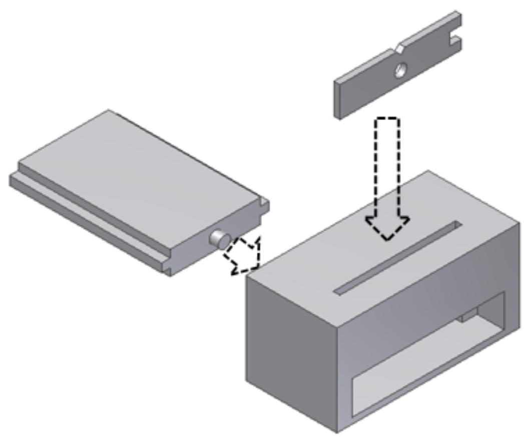


Figure 5.17: Design scenario used for Lab Unit 2.

In order for the blank to be used in the manufacturing process, it must pass certain quality requirements, found in Appendix N. While the basic engineering drawing of the design may seem quite simple, the emphasis in the example is placed on transforming the design requirements into either datums or geometric tolerances. By the end of the exercise, the basic engineering drawing is updated to a fully toleranced print, complete with datums and geometric tolerances, all representing the different requirements placed on the design. This example demonstrates how design requirements can be communicated through geometric tolerances, and how the final product can be interpreted by any individual trained in GD&T, without any additional explanation from the designer.

5.3.2 Lab Unit 2 Assignment A-Practice Reflecting Design Intent through Geometric Tolerances

The assignment for this laboratory is another design scenario, complete with a basic engineering drawing and design requirements. The scenario can be seen in Figure 5.18.

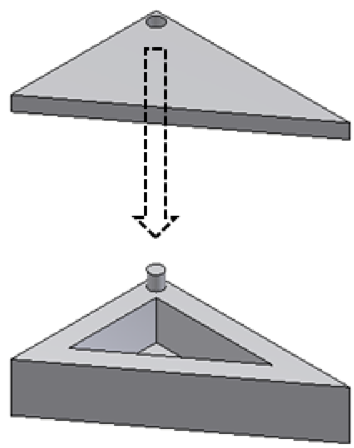


Figure 5.18: Design scenario 2.

The design requirements must then be reviewed and translated into a fully toleranced engineering drawing. An accompanying worksheet also must be completed. This worksheet contains questions that ask which design requirements relate to which tolerances on the drawing. This assignment further exercises the geometric dimensioning and tolerancing skillset and expands it to real-world application.

CHAPTER 6

CONCLUSIONS

Geometric dimensioning and tolerancing is a fundamental engineering tool used in many industries that allows different groups to communicate effectively and efficiently. Due to the importance of GD&T, it is essential for new engineers entering industry to be aware of the tool and if possible develop some level of proficiency. Unfortunately, due to its complex rule based nature, the topic is in general only briefly mentioned or not covered at all in academia. The goal of this thesis is to present the GD&T using a visually based method, and then to provide opportunities to practice the material. As an added benefit, additional tools are incorporated into the program so that multiple topics can be covered simultaneously.

Specifically, the program uses a portable CMM, parametric modeling software, and a three-dimensional printer to simulate a manufacturing and quality inspection scenario. A series of parts are created with manufacturing variation and students are asked to measure these parts using a variety of techniques. Through these different techniques, students are able to understand the importance of datums and also the difference between standard dimensional tolerances and geometric tolerances. The laboratory exercises walk students through four kinds of geometric tolerances, including: profile, orientation, form, and position, and then demonstrates different variational controls within each tolerance class. The final set of laboratory exercises then provide students with a list of functional requirements, which must then be translated into a datum scheme and a fully toleranced drawing. By using an interactive format, the goal of these exercises is to present the material in a stimulating matter so that students can learn the basic aspects of GD&T.

Although geometric dimensioning and tolerancing is not a new topic, it continues to gain more emphasis in industry, as companies attempt to save on manufacturing costs, and to become more environmentally friendly through the elimination of manufacturing waste. While the topic is a difficult one to teach in academia, engineering programs should not underestimate its importance or avoid including it in their curriculum. While the traditional rule-based method for teaching GD&T can be successful, it is not always the most time effective and does not demonstrate to students how it is used in industry. Continued development of these types of interactive programs will not only allow students to be exposed to GD&T, but will also give them an idea of why the topic is important and how it is used in industry. This will allow schools to continue to provide a service that has a direct benefit to both students and their employers.

CHAPTER 7

RECOMMENDATIONS

GD&T is an immense topic and this project is only designed to provide a brief overview of geometric tolerances and how they are used. However, there are a few aspects of this program that can be expanded upon to deliver even more benefit to student.

7.1 Implementation

Before any material can be added to the laboratory exercises, the program should be implemented within a course and data should be gathered to see how well students grasp the material. The program is designed to be presented in a lecture-lab-lecture-lab format. The first lecture component of the curriculum is designed to cover an introduction to GD&T, and also to provide information about the CMM, surface table, and the integration of this equipment with Autodesk Inventor. The first lab exercise is then planned around this information and is aimed at testing student knowledge of GD&T. Through this exercise, it should become clear to what level students are grasping the concepts of GD&T. The only drawback to this approach is that these exercises are suggested to be completed in groups. So one individual could potentially carry the entire group and an accurate reading of each members understanding would not be possible.

The second lecture section is intended to cover design intent and the implementation of GD&T. The lecturer will step through a list of engineering requirements, and demonstrate how to convert these requirements into geometric tolerances and a datum scheme. This exercise is meant to show how to convey design intent through an engineering drawing and also to reinforce GD&T concepts. To test student knowledge, an assignment has been created that simulates the design scenario introduced in the classroom. Through these two tasks, the instructor should be able to gauge how well students grasp the concept.

Additionally, questions can be included on future exams that cover some of the concepts introduced here, and this will provide another avenue for data collection to evaluate the effectiveness of the program. To get direct feedback from students, a survey could be given to assess their level of comfort with GD&T. Through this feedback, the effectiveness of the original program can be judged.

7.2 Expansion

While one geometric tolerance type from each category (except for runout) is covered in detail in the material, other tolerance types could also be included. This can be done through a series of questions on assignments, where students are evaluated on information provided in class. Additionally, more lecture time could be dedicated to GD&T and the remaining tolerances could be covered in detail during class.

Another avenue for expansion is to cover datums and material modifier conditions in more detail. In the first lab background section, datums are only attached to surfaces. However, one of the more convenient aspects of datums is their ability to turn features such as holes, cylinders, or surface sets into datum references. Maximum and minimum material modifiers can also be introduced and covered during the true position tolerance discussion. Finally, tolerance refinement is another avenue for expansion. Through this technique, students can learn how to control angularity, position, and size independently and expand their toolsets.

7.3 Updating

The last area for continual improvement is to update the program as time passes. As the Y14.5M standard for Dimensioning and Tolerancing is updated, the material should be checked to ensure accuracy. Most likely the material is at a basic enough level that standard updates would not affect the program, but due diligence should still be performed. Additionally, as the interfaces for the software change (Microsoft Excel, Autodesk Inventor, HighRES), the screenshots in the program should also be updated to match the updated software.

APPENDIX A

INSTRUCTION SET: PORTABLE CMM SYNCHRONIZATION WITH MICROSOFT EXCEL

The purpose of this instruction set is to synchronize the MicroScribe G2X with Microsoft Excel using the HighRES plug in.

1. Ensure the HighRES USB flash drive with purple key chain is connected to the front USB port in the computer.

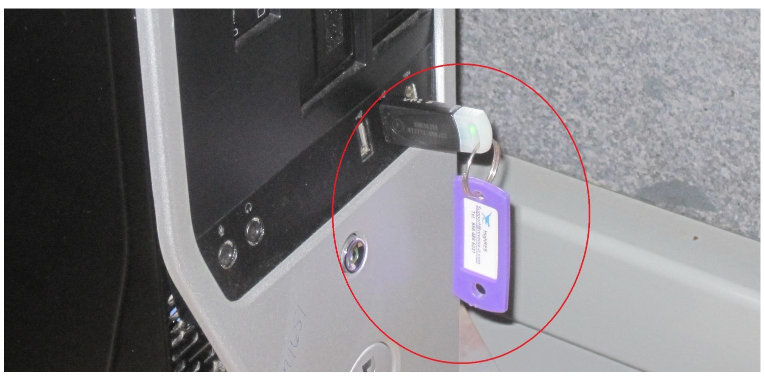


Figure A.1: HighRES USB flash drive with purple key chain.

2. There are four cords that should be connected to the MicroScribe G2X equipment

   - USB cord to the USB drive in the computer
   - 5V DC to an outlet
   - Serial to a 9 pin port in the back of the computer
   - Accessory cable to either the foot pedal or the button pad

3. Navigate to the *Start Menu* ⇒ *All Programs* ⇒ *Immersion Corporation* ⇒ *MicroScribe Utility Software* ⇒ *MicroScribeUtility*, as in Figure A.2. The MicroScribe Utility software dialog box will open.

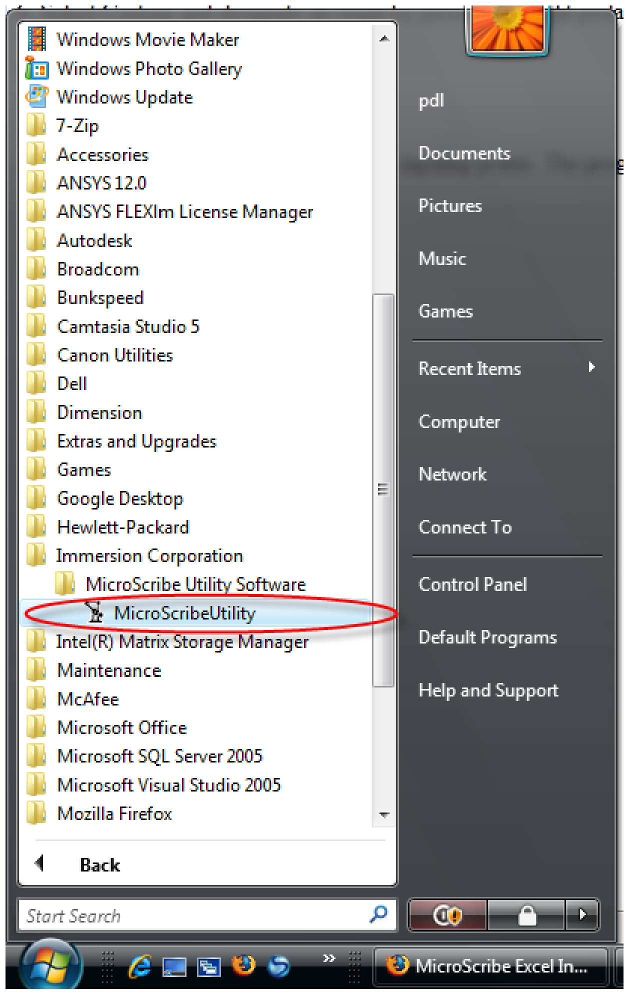


Figure A.2: Double click on the MicroScribe Utility.

4. Within the MicroScribe Utility click on the format strings button. This is the top left button as shown in Figure A.3.

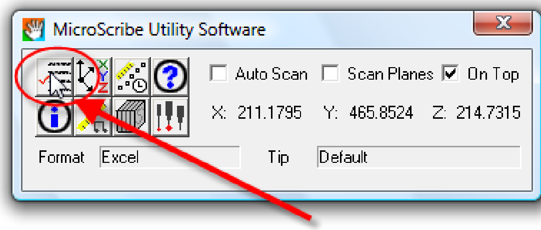


Figure A.3: Click on the format strings option.

5. Verify the *Format Strings* tab is active and select the Excel Format, as shown in Figure A.4.

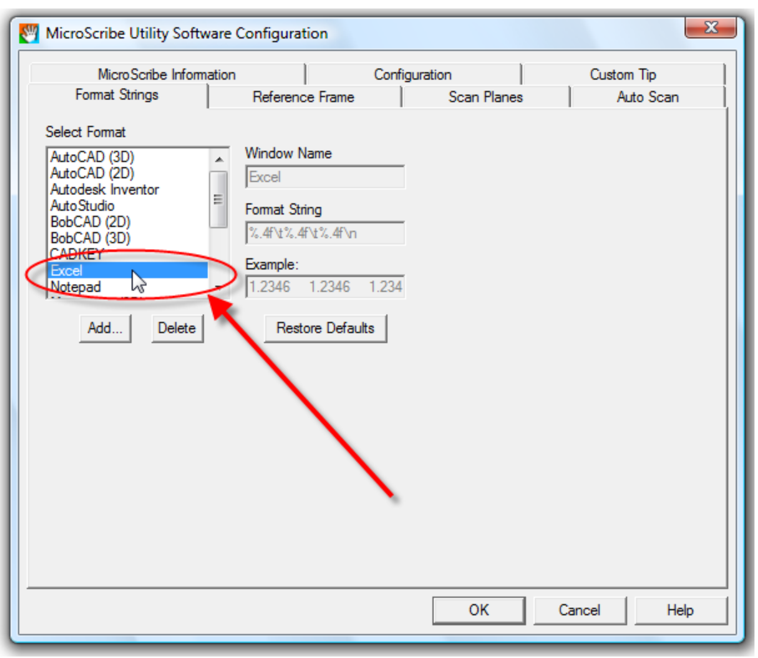


Figure A.4: Select the Excel format.

6. Navigate to the *Reference Frame* tab.

7. Select the Custom option, seen in Figure A.5.

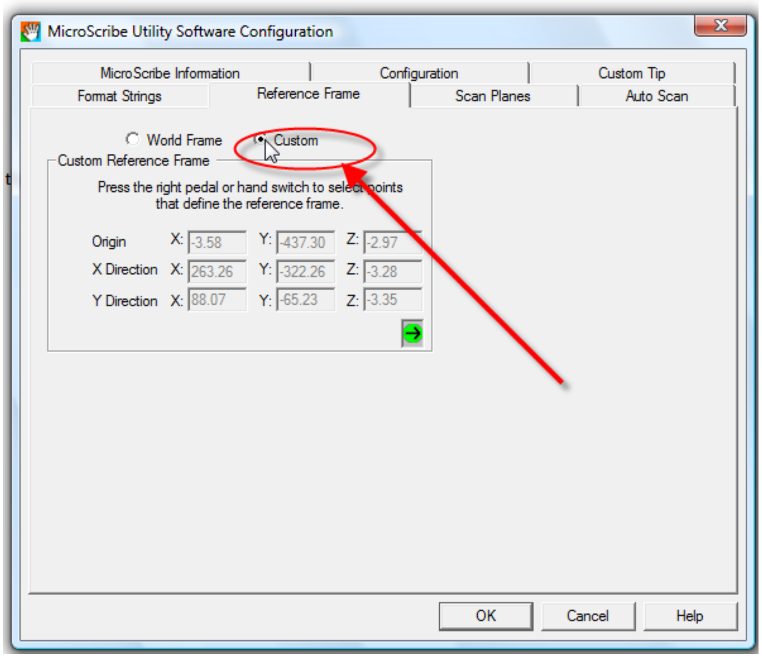


Figure A.5: Select the custom option.

8. Switch to the MicroScribe G2X.

9. Place the CMM measurement tip on the origin of the measurement table and depress the right pedal, as seen in Figure A.6.

(a) Place the tip at the origin.

(b) Depress the right foot pedal.

Figure A.6: Locating the origin.

10. Next touch a calibration point on the granite table and the side measurement plane, as in Figure A.7. **THIS POINT SHOULD BE AS CLOSE TO THE END OF THE INSPECTION RAIL AS POSSIBLE. THIS WILL PRODUCE A LARGE CALIBRATION AREA AND MORE ACCURATELY CAPTURE THE NATURAL SLOPE OF THE ROOM.**

(a) Place the tip on the side measurement rail. This locates the X axis.

(b) Depress the right foot pedal.

Figure A.7: Locating the X axis.

11. Finally, touch a third calibration point on the granite table and the top measurement rail, as in Figure A.8. **THIS POINT SHOULD BE AS CLOSE TO THE END OF THE INSPECTION RAIL AS POSSIBLE. THIS WILL PRODUCE A LARGE CALIBRATION AREA AND MORE ACCURATELY CAPTURE THE NATURAL SLOPE OF THE ROOM.**

(a) Place the tip on the top measurement rail. This locates the Y axis.

(b) Depress the right foot pedal.

Figure A.8: Locating the Y axis.

Note: setting the digitizer matches the coordinate system in Inventor to the measurement table coordinate system. For example, the granite XY plane will match the XY plane in Inventor.

12. The MicroScribe Utility Software Configuration box should now have check marks next to the Origin, X Direction, and Y Direction. This can be seen in Figure A.9.

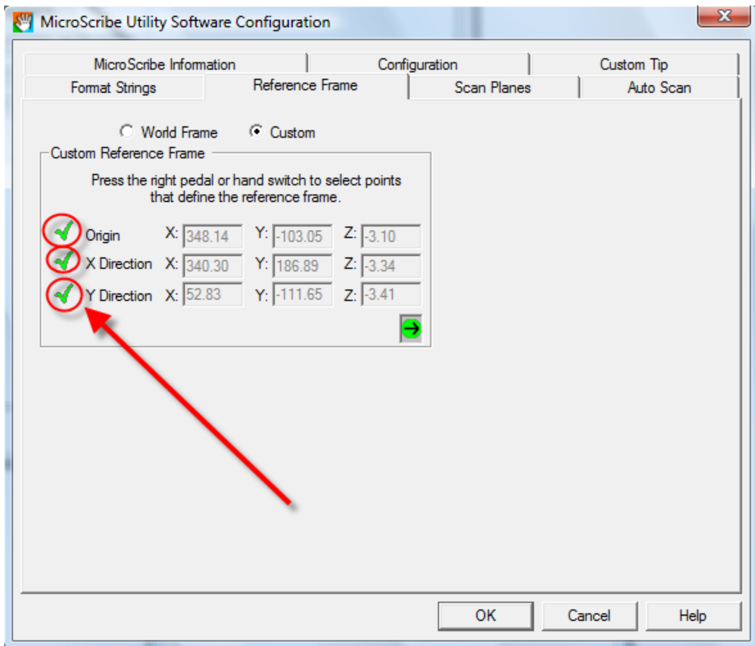


Figure A.9: Green checkmarks next to the Origin, X Direction, and Y Direction.

13. Click OK on the bottom right to close the MicroScribe Utility Software Configuration Dialog Box.

14. **Leave the MicroScribe Utility Software Open!**

15. Open Microsoft Excel.

16. Select a cell to begin taking data.

17. When a point of interest has been found, depress the right foot pedal. Microsoft Excel will automatically record the coordinates of this point and move the input cell down one row.

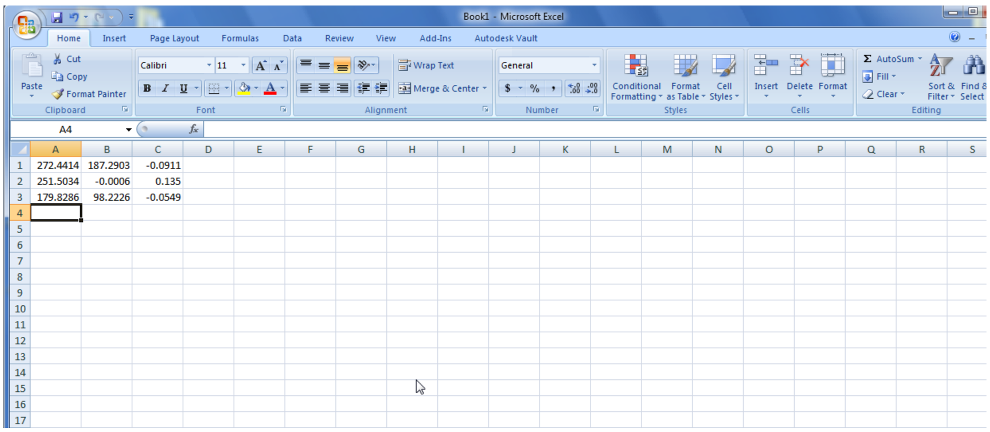


Figure A.10: The MicroScribe G2X reads data directly into Microsoft Excel.

18. Continue this process until all the necessary data is taken.

APPENDIX B

INSTRUCTION SET: PORTABLE CMM SYNCHRONIZATION WITH AUTODESK INVENTOR

The purpose of this instruction set is to synchronize the MicroScribe G2X with Autodesk Inventor using the HighRES plug in.

The first eight steps of this instruction set only need to be completed once during an Autodesk Inventor session. However, steps 9 through 17 will need to be completed for each new part opened.

1. Ensure that the HighRES USB flash drive with purple key chain is connected to the front USB port on the computer. This can be seen in Figure B.1.

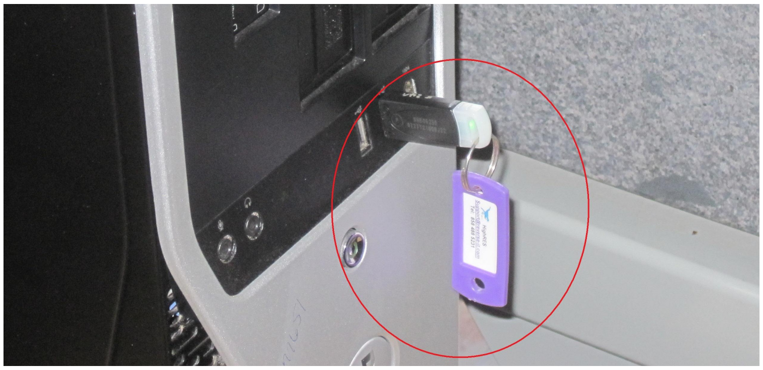


Figure B.1: HighRES USB flash drive with purple key chain.

2. There are four cords that should be connected to the MicroScribe G2X equipment

   - USB cord to the USB drive in the computer
   - 5V DC to an outlet
   - Serial to a 9 pin port in the back of the computer
   - Accessory cable to either the foot pedal or the button pad

3. Navigate to the *Start Menu* ⇒ *All Programs* ⇒ *Immersion Corporation* ⇒ *MicroScribe Utility Software* ⇒ *MicroScribeUtility*, as in Figure B.2. The MicroScribe Utility software dialog box will open.

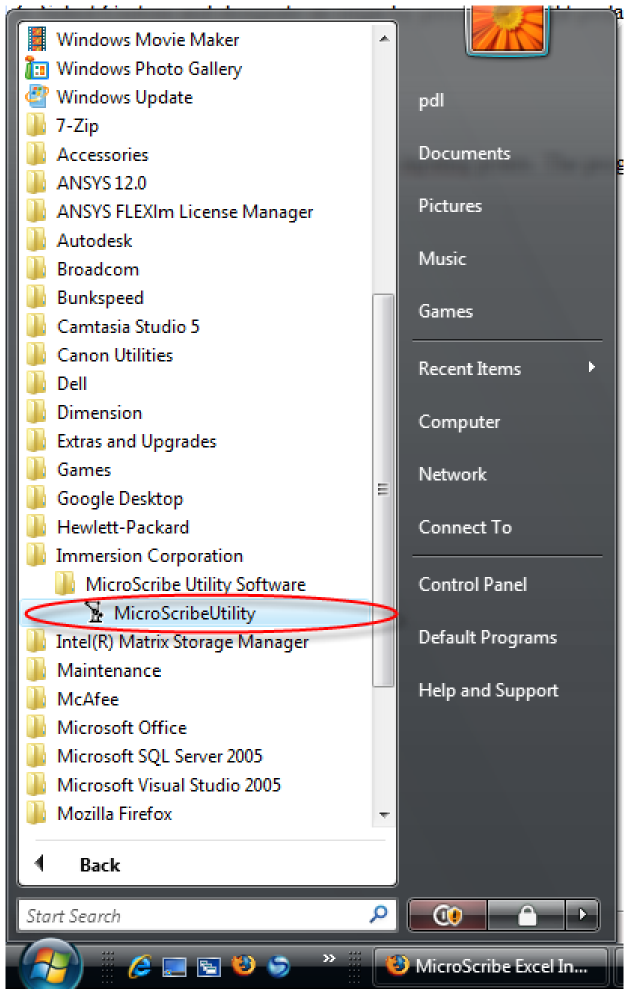


Figure B.2: Double click on the MicroScribe Utility.

4. Within the MicroScribe Utility click on the format strings button. This is the top left button as shown in Figure B.3.

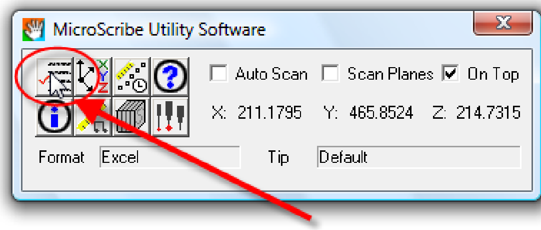


Figure B.3: Click on the format strings option.

5. Verify the *Format Strings* tab is active and select the Autodesk Inventor Format.

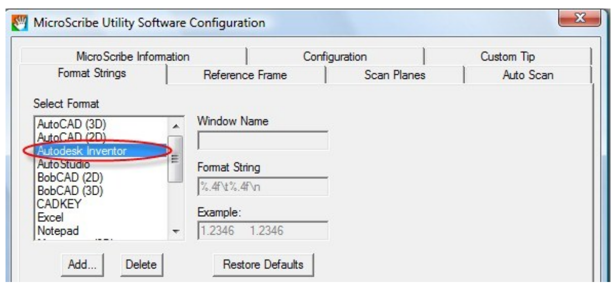


Figure B.4: Select the Autodesk Inventor format.

6. On the Desktop, click on the HIPP 2009 Icon. See Figure B.5.

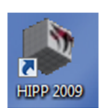


Figure B.5: HIPP 2009 Icon.

7. Navigate to *Toolbox* ⇒ *Connect Scan Device* ⇒ *MicroScribe.*

8. The status box will update and confirm that the device is connected. Minimize the HIPP interface, but do not close.

9. If not done already, start Autodesk Inventor and open the part to be inspected.

10. On the top toolbar click on **CustomUIAddInServer** tab, seen in Figure B.6.

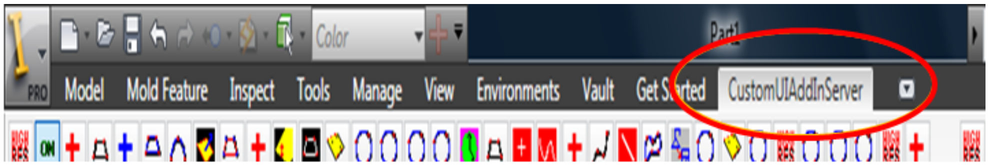


Figure B.6: Click on the CustomUIAddInServer Tab.

11. Click on the Green ON button towards the left. See Figure B.7. A box should pop up that states 'Digitizer has been loaded'.

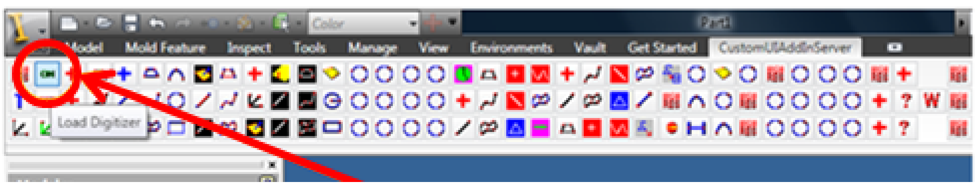


Figure B.7: Click the load the digitizer button.

12. Click on the **Set Digitizer** Icon on the left, as seen in Figure B.8. **AT THIS POINT INVENTOR MAY LOOK FROZEN. IT IS NOT. INVENTOR IS WAITING FOR INPUT FROM THE CMM. DO NOT FORCE QUIT INVENTOR.**

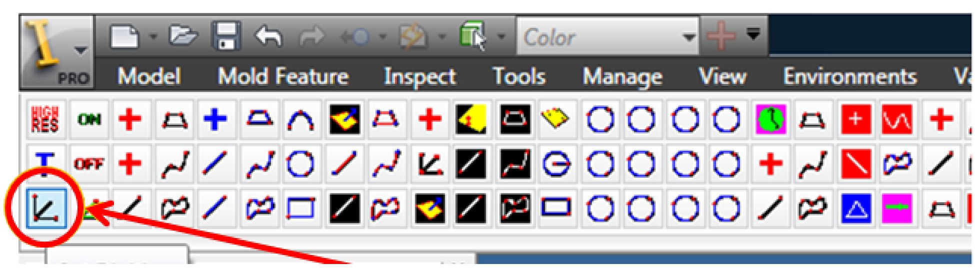


Figure B.8: Click the set digitizer button.

13. Place the CMM measurement tip on the origin of the measurement table and depress the right pedal, as in Figure B.9.

(a) Place the tip at the origin.

(b) Depress the right foot pedal.

Figure B.9: Locating the origin.

14. Next touch a calibration point on the granite table and the side measurement plane, as seen in Figure B.10. **THIS POINT SHOULD BE AS CLOSE TO THE END OF THE INSPECTION RAIL AS POSSIBLE. THIS WILL PRODUCE A LARGE CALIBRATION AREA AND MORE ACCURATELY CAPTURE THE NATURAL SLOPE OF THE ROOM.**

(a) Place the tip on the side measurement rail. This locates the X axis.

(b) Depress the right foot pedal.

Figure B.10: Locating the X axis.

15. Finally, touch a third calibration point on the granite table and the top measurement rail, terminating the Set Digitizer tool. See Figure B.11. **THIS POINT SHOULD BE AS CLOSE TO THE END OF THE INSPECTION RAIL AS POSSIBLE. THIS WILL PRODUCE A LARGE CALIBRATION AREA AND MORE ACCURATELY CAPTURE THE NATURAL SLOPE OF THE ROOM.**

(a) Place the tip on the top measurement rail. This locates the Y axis.

(b) Depress the right foot pedal.

Figure B.11: Locating the Y axis.

- Note: setting the digitizer matches the coordinate system in Inventor to the measurement table coordinate system. For example, the granite XY plane will match the XY plane in Inventor.

16. Right click on the Inventor background (make sure not to right click on a surface or plane) and select the **Finish 3D Sketch**.

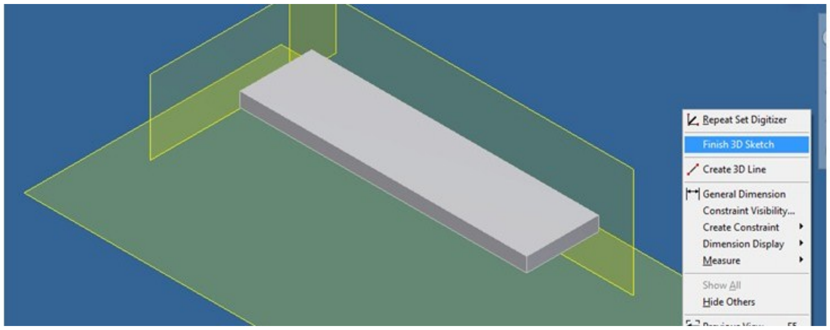


Figure B.12: Right click in the workspace and select Finish 3D sketch.

17. The CMM is now interfaced with Inventor and the two coordinate systems are aligned. IF THE CMM BASE IS MOVED OR A NEW PART IS OPENED IN INVENTOR, THE CMM WILL NEED TO BE RECALIBRATED. THIS PROCESS STARTS WITH STEP 7.

Table B.1: Coordinate Alignment

Autodesk Inventor		Measurement Table
YZ Plane	⇔	Top Rail
XZ Plane	⇔	Side Rail
XY Plane	⇔	Granite Surface

APPENDIX C

INSTRUCTION SET: TAKING SURFACE INSPECTION POINTS

The purpose of this instruction set is to utilize the MicroScribe G2X and Autodesk Inventor to take inspection points. Once the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* (see Appendix B) has been completed and the part of interest is fixtured on the measurement table, the next step is to take inspection points.

1. On the left side of the CMM user interface, click on the 3D point button. This button is located directly to the right of the ON button, as shown in Figure C.1. **AUTODESK INVENTOR WILL APPEAR FROZEN. THIS IS NORMAL, IT IS WAITING FOR INPUT FROM THE CMM. DO NOT FORCE QUIT THE APPLICATION. IF YOU HAVE COMPLETED TAKING POINTS, SEE STEP 4 FOR HELP.**

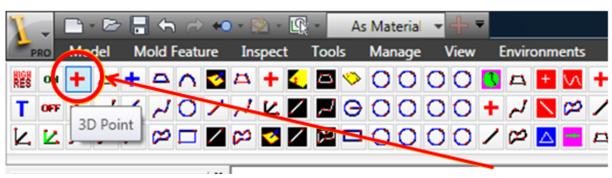


Figure C.1: Click on the 3D point button.

2. Place the CMM tip on the first measurement point, as seen in Figure C.2(a), and depress the right foot pedal, as seen in Figure C.2(b). The first point has now been taken.

(a) CMM tip on the first measure point.

(b) Depress the right foot pedal.

Figure C.2: Depress right foot pedal to record data point.

3. Repeat this process for the remaining points.

4. When all points have been taken depress the left foot pedal. The left foot pedal is used to end a command such as the 3D Point tool. This will signal to Inventor that all input from the CMM has stopped and it can resume functioning. This process will also unfreeze Inventor.

Figure C.3: Depress left foot pedal to exit point recording.

5. The measurement points should now appear near the surface of the part in Autodesk Inventor as seen in Figure C.4.

   Note: Some of the points may be hidden by the surface of the block. If this is the case, click on the *View Tab* in Autodesk Inventor and change the shading to *Wireframe* to see these points.

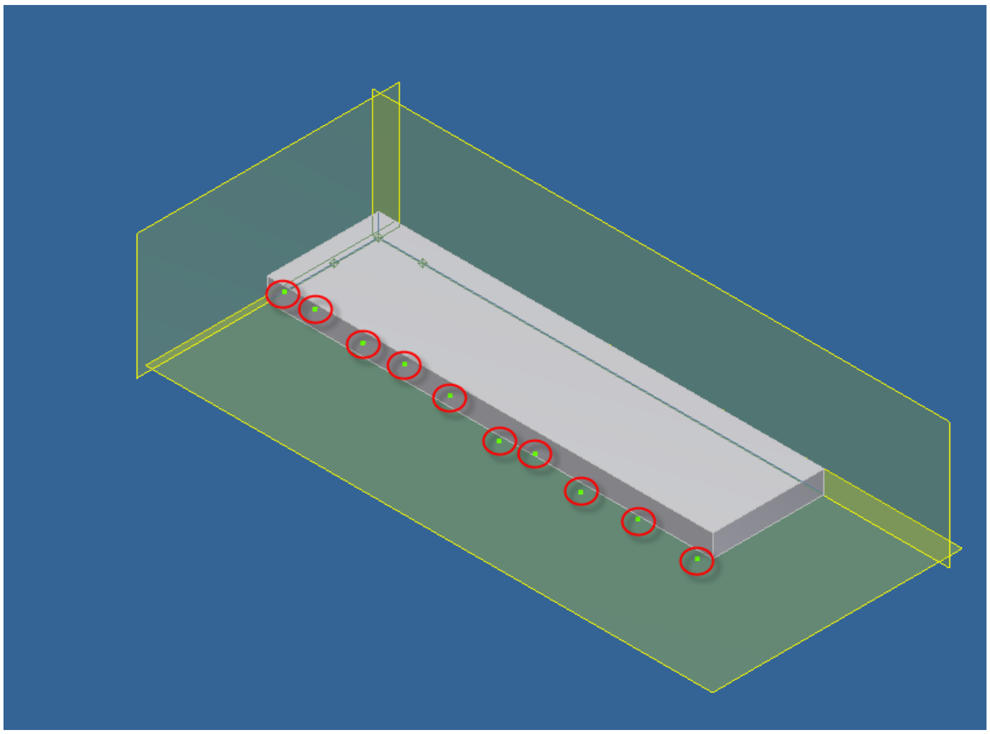


Figure C.4: Points taken from the CMM.

APPENDIX D

INSTRUCTION SET: TRUE POSITION INSPECTION POINTS

The purpose of this instruction set is to utilize the MicroScribe G2X and Autodesk Inventor to take true position inspection points. Once the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* (see Appendix B) has been completed and a part of interest with a cylindrical feature is fixtured on the measurement table, the next step is to take inspection points.

1. Under the *CustomUIAddInServer* select the *3 Point 2D Circle* as seen in Figure D.1.

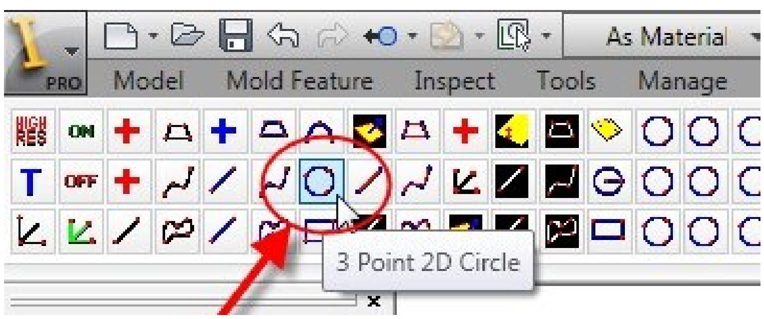


Figure D.1: Select the 3 Point 2D Circle.

2. Select one point on the base circle and depress the right foot pedal to record the data point as in Figure D.2.

(a) Place the probe on the first inspection point. (b) Depress the right foot pedal to register the point.

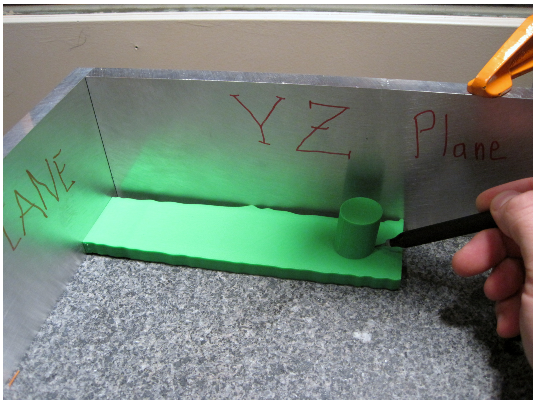
Figure D.2: Taking the first inspection point to form the 3 point circle.

3. Select a second point on the base circle and depress the right foot pedal to record the data point as in Figure D.3. Make sure this point is about one third around the circumference from the first point.

(a) Place the probe on the second inspection point. (b) Depress the right foot pedal to register the point.

Figure D.3: Taking the second inspection point to form the 3 point circle.

4. Select a third point on the base circle and depress the right foot pedal to record the data point as in Figure D.4. Make sure this point is evenly spaced between the first point and the second point.



(a) Place the probe on the third inspection point.

(b) Depress the right foot pedal to register the point.

Figure D.4: Taking the third inspection point to form the 3 point circle.

5. Under the *View* tab in Autodesk Inventor, change the shading to *Wireframe*.

6. Use the *Navigation Cube* and select the *Top View*. The Autodesk Inventor screen should look similar to Figure D.5.

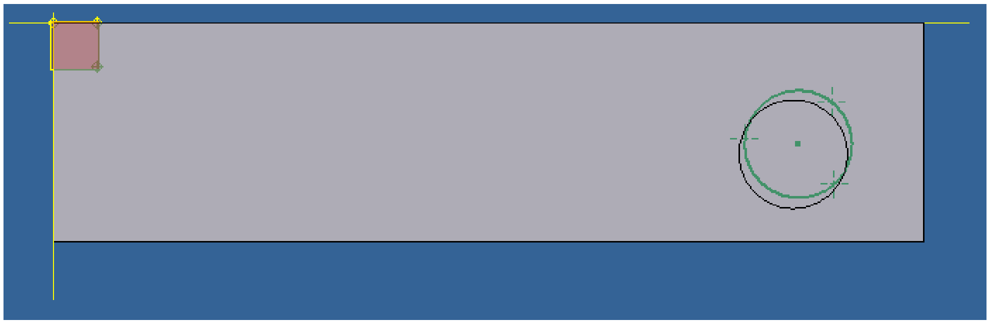
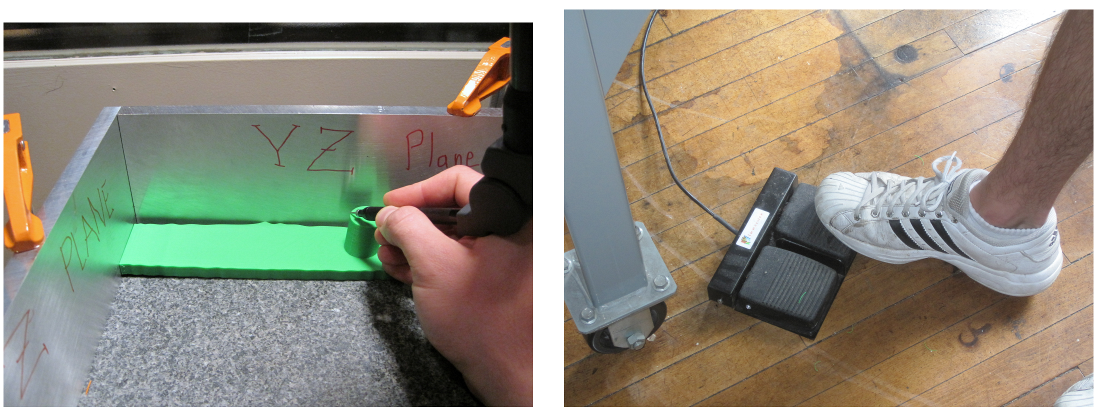


Figure D.5: Top view of the base circle.

7. Again under the *CustomUIAddInServer* select the *3 Point 2D Circle* function.

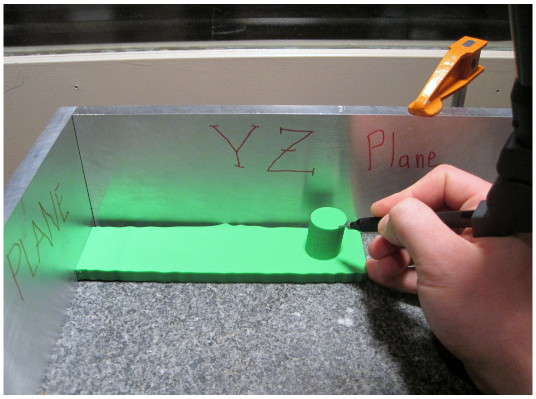
8. This time select one point on the top circle and depress the right foot pedal to record the data point as in Figure D.6.



(a) Place the probe on the first inspection point. (b) Depress the right foot pedal to register the point.

Figure D.6: Taking the first inspection point to form the 3 point circle.

9. Select a second point on the top circle and depress the right foot pedal to record the data point as in Figure D.7. Make sure this point is about one third around the circumference from the first point.

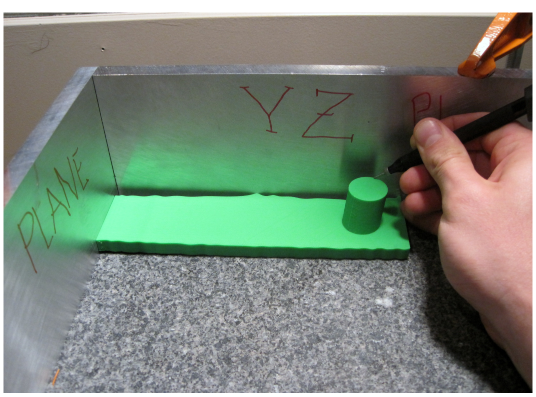


(a) Place the probe on the second inspection point.

(b) Depress the right foot pedal to register the point.

Figure D.7: Taking the second inspection point to form the 3 point circle.

10. Select a third point on the top circle and depress the right foot pedal to record the data point as in Figure D.8. Make sure this point is evenly spaced between the first point and the second point.



(a) Place the probe on the third inspection point.

(b) Depress the right foot pedal to register the point.

Figure D.8: Taking the third inspection point to form the 3 point circle.

11. The Autodesk Inventor screen should be similar to Figure D.9. Note how the two inspection circles lie on the base XY plane.

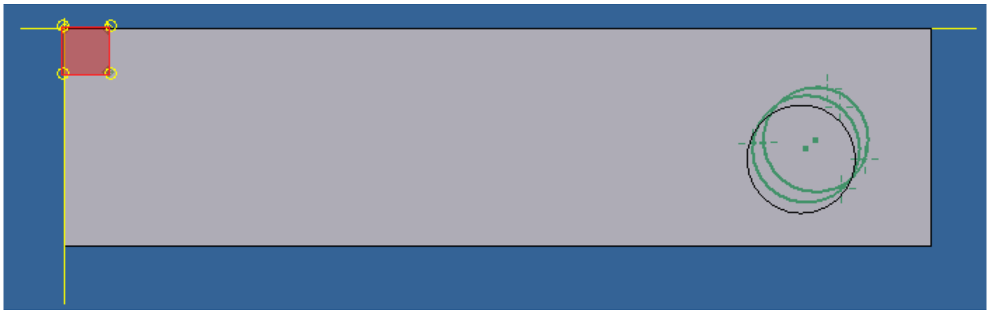


Figure D.9: Top view of the two inspection circles.

APPENDIX E

LAB UNIT ONE-LECTURE: GD&T LAB BACKGROUND INFORMATION

This background information should be reviewed and understood before attempting to complete the accompanying laboratory. This section will give a basic introduction to geometric dimensioning and tolerancing and explain the concepts behind the form, orientation, profile, and location tolerance classes.

E.1 Datums

The first step in describing the function of a part on an engineering drawing is to identify the important features on a part and which features need to be related to others based on part function. From these important features a system of baselines can be established, which help communicate to others how to manufacture, inspect, and measure a part. In an engineering drawing, these baselines are known as datums.

E.1.1 Datums Establish a Coordinate System for Measurement through Datum Reference Frames (DRF's)

In the real world, all parts are three dimensional. A part can be defined in space, by describing where every point on the part is located with coordinates. The most familiar system for mapping points this way is the Cartesian coordinate system, which uses an X, Y, and Z plane to establish the coordinates of a point.

One of the purposes of a datum in an engineering drawing is to specify what surfaces or features on a part will be used to establish the baseline coordinate system, known as a datum reference frame (DRF). Tolerances then reference these datums to establish relationships amoung features. An example of how datums can be represented in an engineering drawing can be seen in Figure E.1.

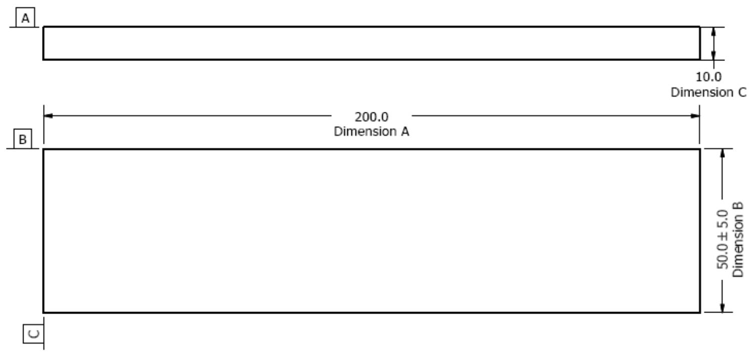


Figure E.1: Engineering drawing of Block 1 with the datums boxed.

In Figure E.1, the ***-A-*** datum is designated as the top surface in the top view. The ***-B-*** datum is designated as the top surface and the ***-C-*** datum is the left surface in the front view. In the case of a rectangular block it is not possible to distinguish what orientation the block should be in when assigning datums. Due to this, either configuration of the block may be used to assign the datum scheme (i.e. either large flat face can be the front face).

Some important points about datums:

- There is no limit to the number of datums that can be added to a drawing. On more complicated parts, the same coordinate system is not always used to define each feature. There may be multiple DRFs on one part, depending on how the features are functionally related.

- The most important functional feature on a part is typically assigned as the ***-A-*** datum. The next feature of importance is given the ***-B-*** datum, and so on. The engineer decides what the most important feature on a part is during the design process. This also enables the engineer to convey design intent.

- It is generally good practice to specify at least three datums on an engineering drawing, so all of the degrees of freedom (Section E.1.3) of the part are controlled and coordinate system is fully defined. However, there are circumstances where only the relevant degrees of freedom need to be controlled on a part and three datums are not required.

E.1.2 The Inspection Table Coordinate System

A coordinate measuring machine, or CMM, also operates by utilizing three planes to establish a coordinate system. In the case of the Product Dissection Laboratory (PDL), the portable CMM is placed on the inspection table, which consists of a granite surface plate and two precision ground aluminum blocks. The granite forms the XY-plane, the top measurement rail forms the YZ-plane, and the side measurement rail forms the XZ-plane. This can be seen in Figure E.2.

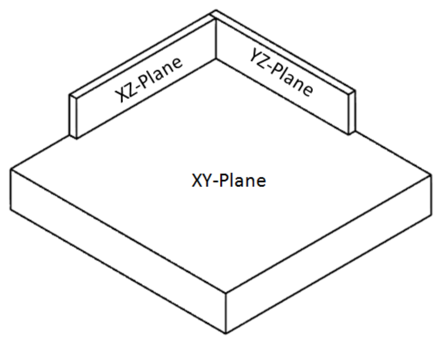


Figure E.2: The Cartesian coordinate system of the inspection table.

E.1.3 Primary, Secondary, and Tertiary Datums

A datum reference frame in an engineering drawing also serves to match the part coordinate system with the inspection table coordinate system for each feature inspected. This is done is by specifying which datums constrain the part to the measurement table.

Just as constraints are used to restrict part freedom in an Autodesk Inventor assembly, datums are used as constraints during an inspection process. It generally takes three datums to fully remove all six degrees of freedom (DOF) of a part (three translations and three rotations). The datums used for constraints are called the primary, secondary, and tertiary datums, because they are applied in that order. A schematic of this concept can be seen in Figure E.3.

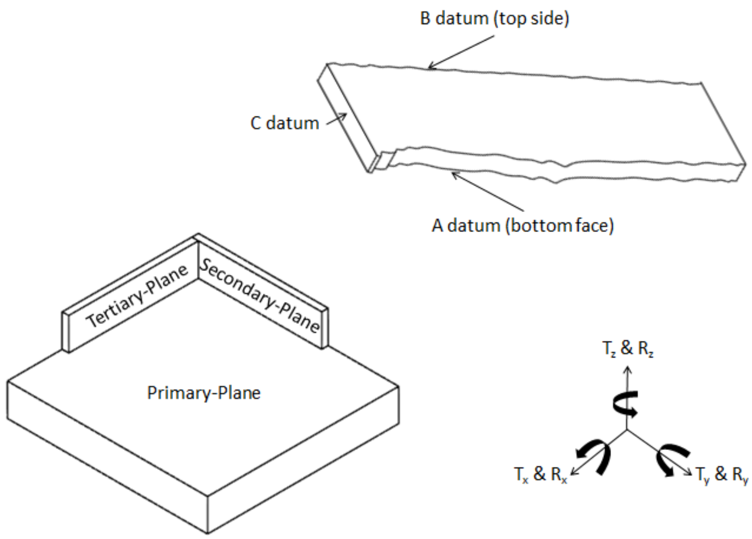


Figure E.3: Degrees of Freedom of Block 1. T represents translations, R represents rotations, and the subscript denotes the axial direction.

The three highest points of the primary datum feature contact the primary measurement plane on the inspection table. These three points of contact are used to define a plane, which constrains two rotations (R_X and R_Y) and a translation (T_Z), but T_X, T_Y, and R_Z are still free. In the case of block 1, the contact between the ***-A-*** datum, or the large face of block, and the granite surface plate creates this planar contact. The block has three remaining degrees of freedom, two translations and one rotation, which can be seen in Figure E.4.

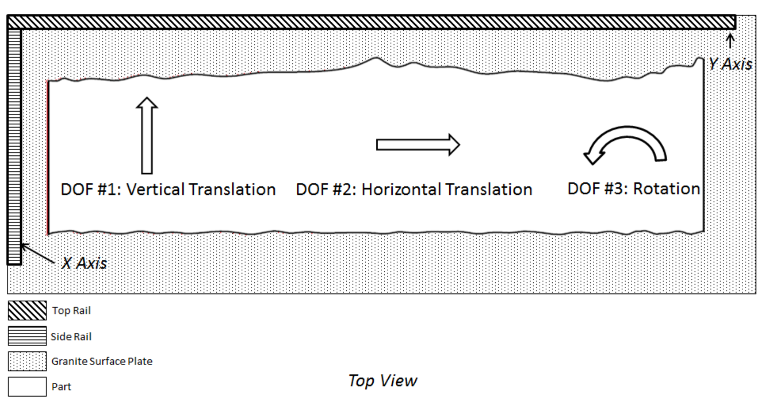


Figure E.4: The Z translation, X rotation, and Y rotation have been removed with the planar contact. The Z rotation, X translation, and Y translation remain.

The secondary datum is intended to have at least two point contacts with the secondary measurement plane, which creates a line contact and removes another rotation and translation from the part. The first of the point contacts will take away the vertical translation of the part and can be seen in Figure E.5.

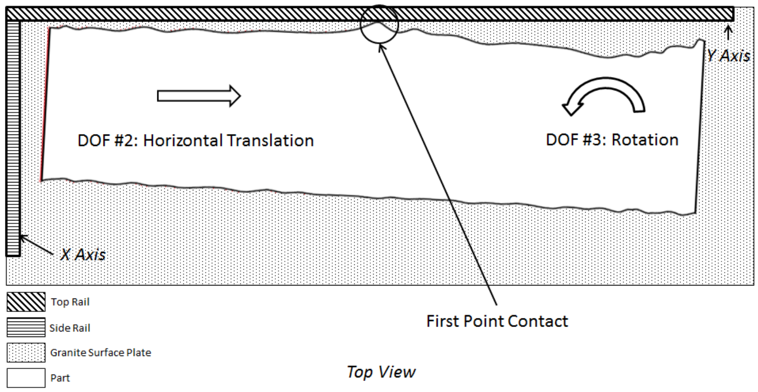


Figure E.5: First contact constrains X translation. Z rotation and Y translation free.

The second point will constrain the rotation. In some cases the block can be rotated in either direction to find a second point contact, but clock-wise rotation was used in this example. These two points will form a line, which lies in the plane of the top measurement rail, and can be seen in Figure E.6.

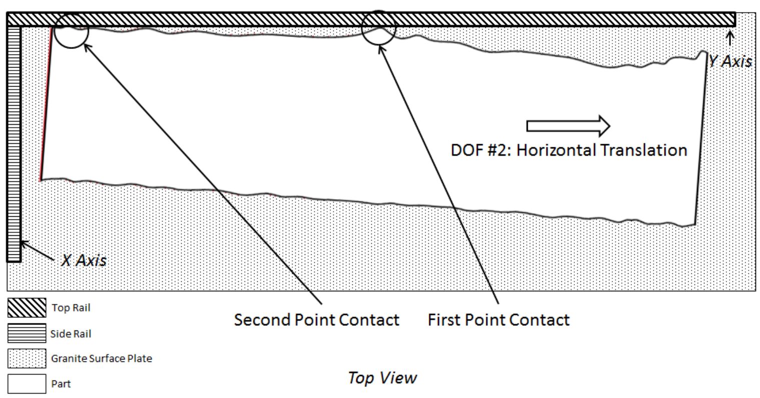


Figure E.6: Second contact constrains Z rotation. Y translation free.

The tertiary datum is intended to have point contact with the tertiary measurement plane, as in Figure E.7. This removes the final translation DOF and the block is fully constrained on the measurement table.

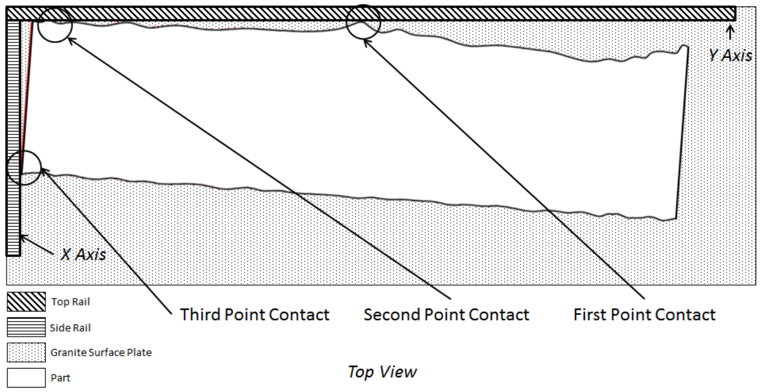


Figure E.7: The final point contact removes the final DOF- the Y translation.

A summary of the DOF removed can be seen in Table E.1. Note: The primary, secondary, and tertiary datums do not always correspond with the ***-A-*** datum, ***-B-*** datum, and ***-C-*** datum.

Table E.1: DOF removed through the primary, secondary, and tertiary datums for Figure E.3

	Primary			Secondary			Tertiary		
Datum	***-A-***			***-B-***			***-C-***		
DOF Removed	T_Z	R_X	R_Y	R_Z	T_X	-	T_Y	-	-

E.2 Geometric Tolerances

A geometric tolerance, much like a dimensional tolerance, is used in an engineering drawing to describe where a feature needs to be for the part to function correctly. It is important to first identify what a geometric tolerance looks like and its different components.

E.2.1 Feature Control Frame

A geometric tolerance appears in what is known as a feature control frame, which can be seen in Figure E.8.

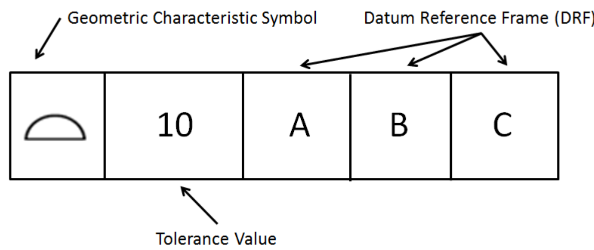


Figure E.8: An example feature control frame.

There are three main sections to a feature control frame:

1. The first entry in the feature control frame is the geometric characteristic symbol. This symbol conveys what tolerance should be applied during manufacturing and checking of the feature. In this case, the geometric tolerance is a profile tolerance but the meaning of the tolerance will be discussed later.

2. The second entry in the feature control frame is the tolerance value, which in this case is 10 mm. This value describes the allowable tolerance zone that the feature can fall within.

3. The final entries in the feature control frame specify the datum order used when inspecting a part, which establish the measurement baselines. In this case, ***-A-*** will be used as the primary datum, ***-B-*** as the secondary datum, and ***-C-*** as the tertiary datum. This is the datum reference frame. However, the order of the features could be re-written as ***-A-***, ***-C-***, ***-B-***, where ***-A-*** would be the primary datum, ***-C-*** would be the secondary datum, and ***-B-*** would be the tertiary datum and the way the part is held for inspection changes.

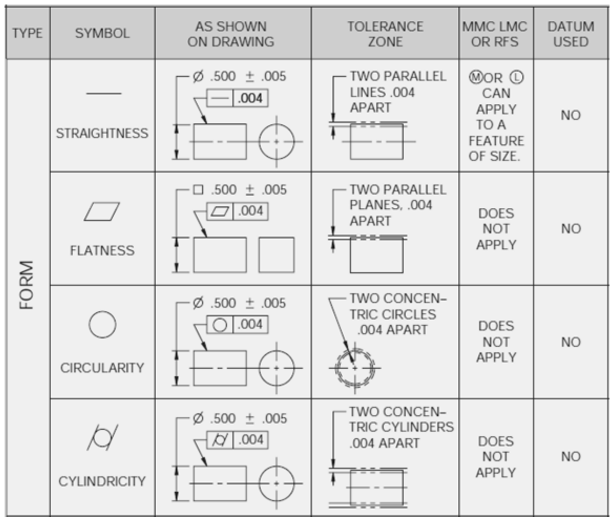
E.2.2 Geometric Tolerance Categories

In geometric dimensioning, there are five main classes of tolerances. They are as follows:

- Form Tolerances (see Figure E.9)
- Orientation Tolerances (see Figure E.10)

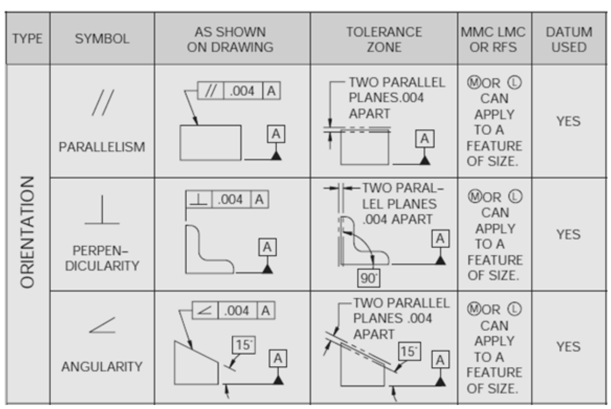
- Profile Tolerances (see Figure E.11)
- Runout Tolerances (not covered)
- Location Tolerances (see Figure E.12

Within these tolerance classes there are different types of tolerances. A summary of the main tolerances can be seen below.



TYPE	SYMBOL	AS SHOWN ON DRAWING	TOLERANCE ZONE	MMC LMC OR RFS	DATUM USED
FORM	— STRAIGHTNESS	Ø .500 ± .005 — .004	TWO PARALLEL LINES .004 APART	Ⓜ OR Ⓛ CAN APPLY TO A FEATURE OF SIZE.	NO
	▱ FLATNESS	□ .500 ± .005 ▱ .004	TWO PARALLEL PLANES, .004 APART	DOES NOT APPLY	NO
	○ CIRCULARITY	Ø .500 ± .005 ○ .004	TWO CONCENTRIC CIRCLES .004 APART	DOES NOT APPLY	NO
	⌭ CYLINDRICITY	Ø .500 ± .005 ⌭ .004	TWO CONCENTRIC CYLINDERS .004 APART	DOES NOT APPLY	NO

Figure E.9: Form Tolerance Class.



TYPE	SYMBOL	AS SHOWN ON DRAWING	TOLERANCE ZONE	MMC LMC OR RFS	DATUM USED
ORIENTATION	// PARALLELISM	// .004 A; A	TWO PARALLEL PLANES.004 APART; A	Ⓜ OR Ⓛ CAN APPLY TO A FEATURE OF SIZE.	YES
	⊥ PERPENDICULARITY	⊥ .004 A; A	TWO PARALLEL PLANES .004 APART; A; 90°	Ⓜ OR Ⓛ CAN APPLY TO A FEATURE OF SIZE.	YES
	∠ ANGULARITY	∠ .004 A; 15°; A	TWO PARALLEL PLANES .004 APART; 15°; A	Ⓜ OR Ⓛ CAN APPLY TO A FEATURE OF SIZE.	YES

Figure E.10: Orientation Tolerance Class.

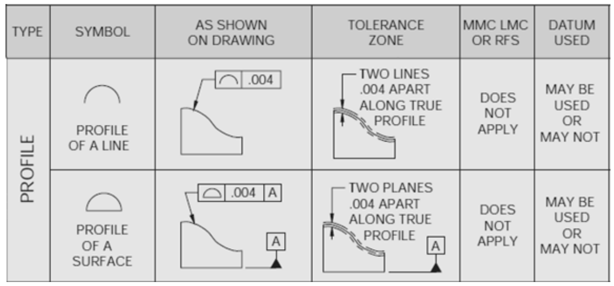


Figure E.11: Profile Tolerance Class.

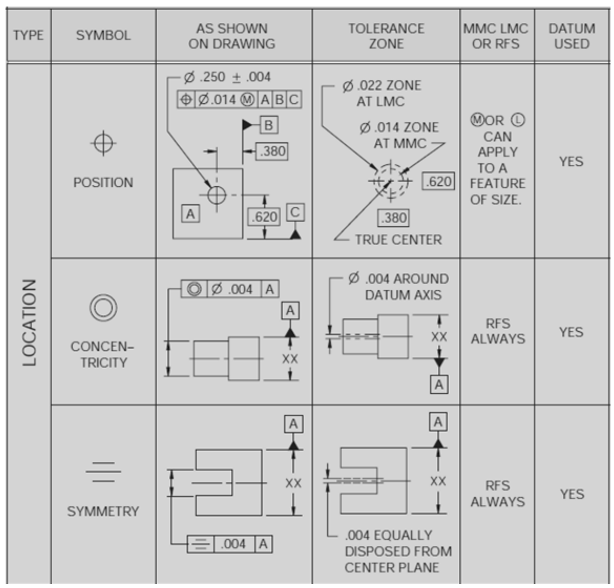


Figure E.12: Location Tolerance Class.

To demonstrate the usage of a geometric tolerance, Block 1 will be used again. This time the engineering drawing has been updated to incorporate a geometric tolerance, which can be seen in Figure E.13.

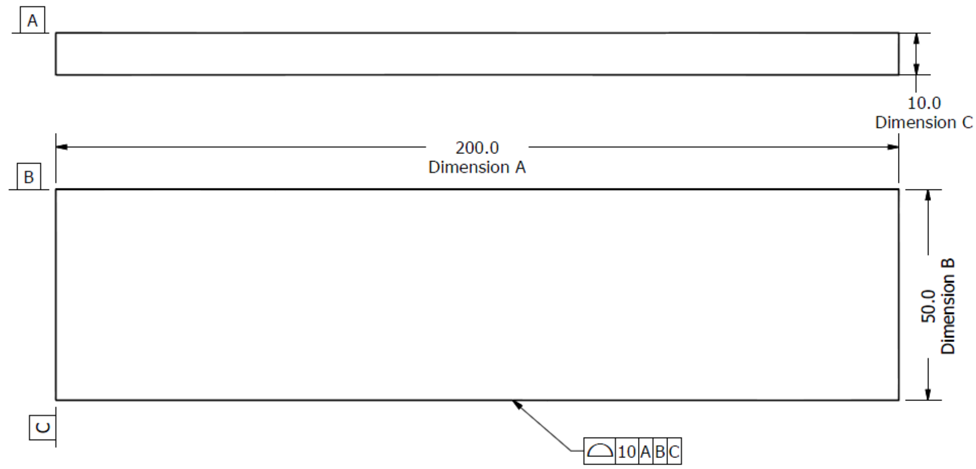


Figure E.13: Drawing of Block 1 with datums and a profile geometric tolerance.

E.2.3 Profile Class: *Profile of a Surface Tolerance* Fundamentals

A profile tolerance of 10 with respect to the ***-A-***, ***-B-***, and ***-C-*** datums has been applied to the surface opposite of the ***-B-*** datum. This profile tolerance falls into the category of a location tolerance, because it will help specify where this surface should be located. In addition to location, the profile tolerance also controls size, orientation, and form. The next few steps will demonstrate how to interpret and understand the function of a profile tolerance.

Step 1: Fixture the Block

The first step to examining a profile tolerance is to recognize what datums are referenced in the feature control frame. The profile tolerance specifies that ***-A-*** is the primary datum, ***-B-*** is the secondary datum, and ***-C-*** is the tertiary datum. Block 1 is then fixtured by matching the primary, secondary, and tertiary datums of the part (specified in the feature control frame) to the simulated primary, secondary, and tertiary datums of the inspection table. The granite surface, top rail, and side rail simulate the ***-A-*** datum, ***-B-*** datum, ***-C-*** datum. This fixturing can be seen in Figure E.14.

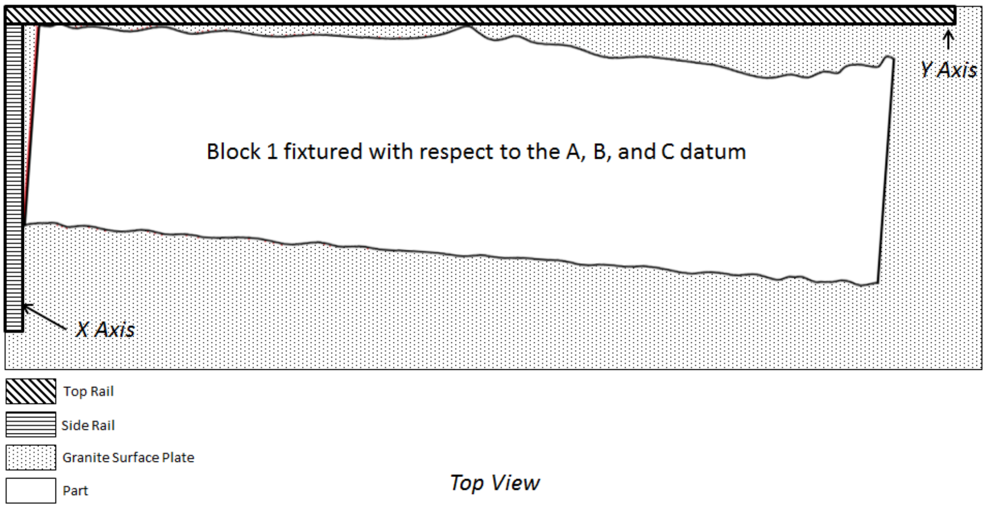


Figure E.14: Block 1 fully fixture to the measurement table.

Step 2: Create Perfect Surface

The next step in understanding the profile geometric tolerance is to identify where the theoretically perfect surface is located with respect to the fixturing datums. According to the engineering drawing, the perfect surface should be located 50 mm from datum ***-B-***, which coincides with the top measurement rail. The height of the perfect surface stops a distance of 10 mm above the ***-A-*** datum, which coincides with the granite surface. Finally, the length of the perfect surface stops 200 mm from the ***-C-*** datum, which coincides with the side measurement rail. This perfect surface is virtual and serves as a starting point to locate the tolerance zone. This perfect plane can be seen in Figure E.15.

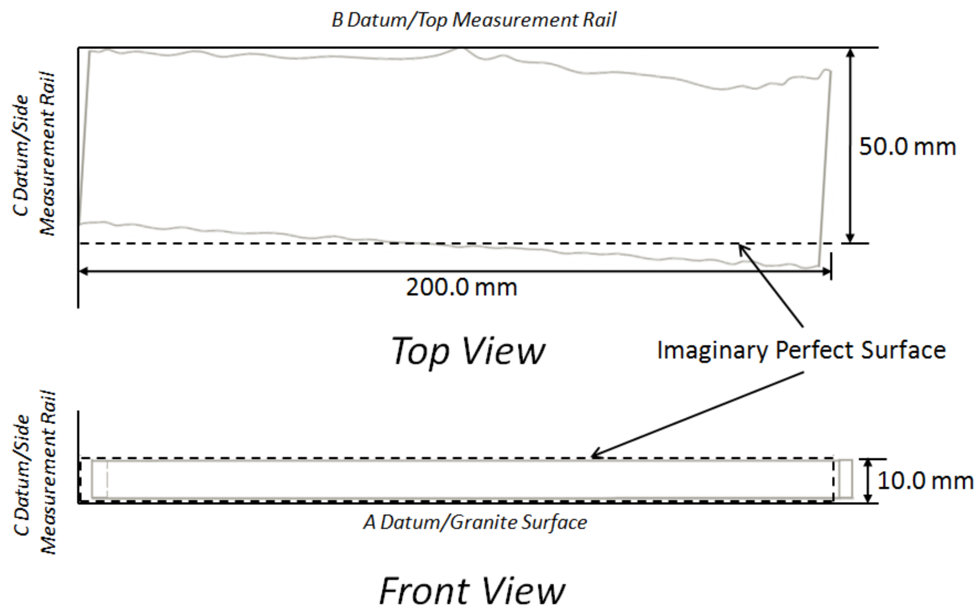


Figure E.15: Establishing the perfect surface on the inspection table.

Step 3: Create the Planar Boundaries

The final step in applying the profile geometric tolerance is to create the tolerance zone around the perfect surface. The boundary for this zone is formed by a pair of planes, which are 10 mm apart (as specified in the feature control frame). The region between these two planes creates a zone, which can be seen in Figure E.16. The actual surface of the block must fit within this zone, but as long as it remains within the bounds it may take any shape (curved, straight, wavy, etc.), any orientation or angle with respect to the measurement planes, or any combination of shape and orientation. In order to check to see if the surface of the Block 1 fits within the boundaries, a series of measurements are taken with the Microscribe G2X. This will be practiced later.

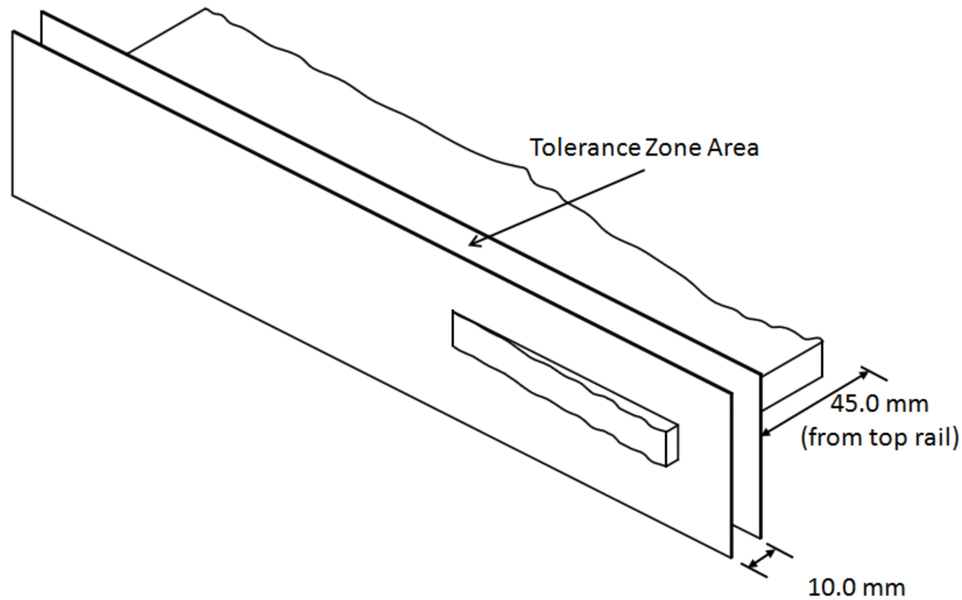


Figure E.16: Establishing the perfect tolerance zone or boundary area.

Geometric Tolerance vs. Dimensional Tolerance

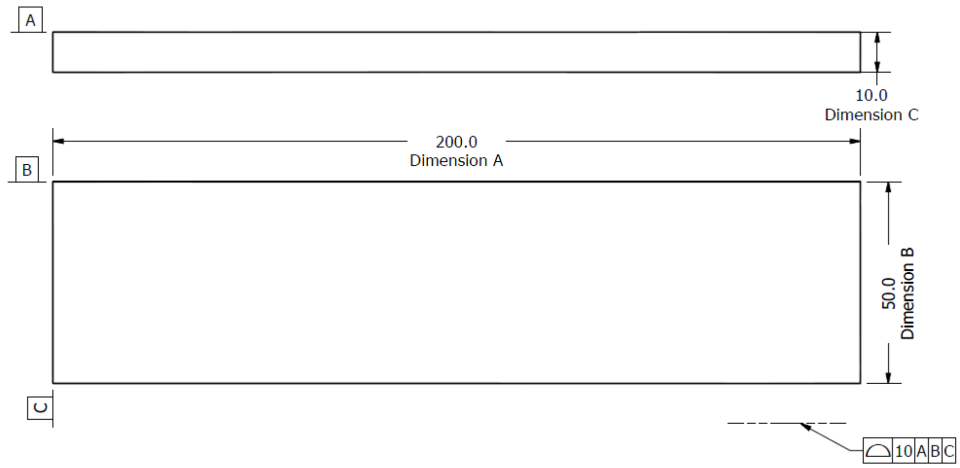
While both geometric tolerances and dimensional tolerances serve to control variation, they differ in the following ways:

- A dimensional tolerance controls the surface in one dimension, while a geometric tolerance can control the surface in one, two, or three dimensions.
- With a dimensional tolerance, the 10 mm variation in the 50.0 mm dimension is shared between the ***-B-*** datum and the surface opposite the ***-B-*** datum. With the geometric tolerance, the two surfaces are allowed vary independently. This will become clear during the lab exercise.
- Since the two surfaces are allowed to vary independently, the designer can now control each surface separately with geometric tolerances. The surface opposite the ***-B-*** datum is now controlled with a profile tolerance, while the ***-B-*** datum surface is currently uncontrolled (this surface will be controlled next).

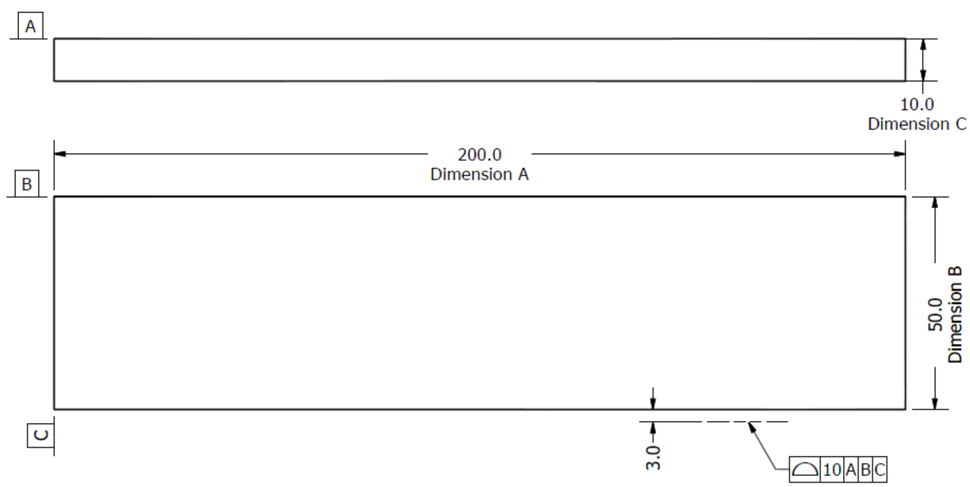
- By controlling the two surfaces separately the engineer knows where exactly the variation is allowed. This will allow for better control of the block and clearly convey design intent.

Profile Tolerance Special Cases: Unilateral and Unequal, Bilateral Zones

A profile tolerance does not always have to be applied symmetrically about the virtual perfect surface. It also can be applied unilaterally, which consists of a zone located on one side of the surface. The other alternative for a profile tolerance boundary may be an unequal, bilateral zone, which consists of an asymmetric zone on either side of the surface. Examples of these can be seen in Figure E.17(a) and Figure E.17(b).



(a) An example of a unilateral profile tolerance. All 10 mm of variation occur to the outside of the perfect surface as indicated by the phantom line.



(b) An example of an unequal, bilateral profile tolerance. 3 mm of variation occur to the outside of the perfect surface, while 7 mm of variation are allowed to the inside the perfect surface. This is indicated by the dimension on the phantom line.

Figure E.17: Special cases of a profile tolerance.

E.2.4 Profile Tolerance Example Inspection

The first step in inspecting a profile tolerance applied to a manufactured block is to take inspection points off of the surface of interest.

Establish Inspection Points

1. Begin by starting Autodesk Inventor and opening `Block_1_CMM_Measurement_Profile.ipt`.

2. Initialize and Calibrate the CMM according to the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* found in Appendix B.

3. Fixture the block to the CMM measurement table using the primary, secondary, and tertiary datums designated in the feature control frame found on engineering drawing.

Figure E.18: Fixturing the block with the primary, secondary, and tertiary datums.

4. Take the necessary CMM measurements along the surface opposite of datum ***-B-*** according to the *Instruction Set: Taking Surface Inspection Points* found in Appendix C. Be sure to keep the Microscribe base and the part still during the entire measurement process.

5. After all the points are taken, the Autodesk Inventor screen should look similar to

Figure E.19.

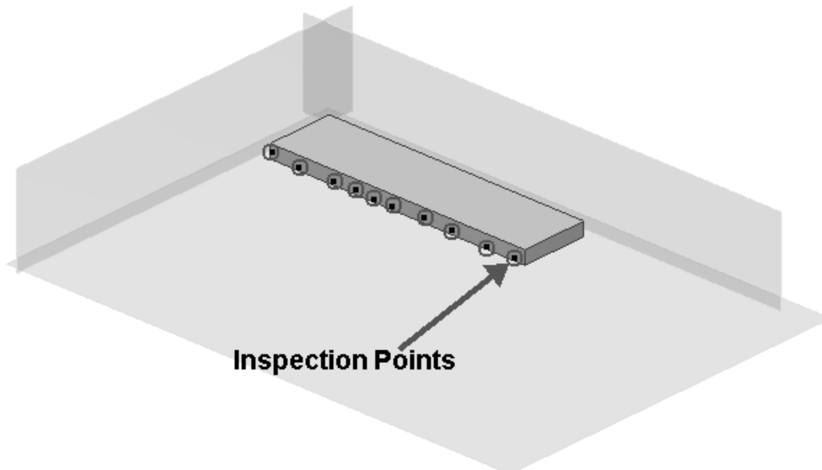


Figure E.19: CMM inspection points in Autodesk Inventor.

Establish the Tolerance Zone

The next step in the inspection process is to establish the perfect surface so a tolerance zone can be created. Since the Autodesk Inventor part file represents the perfect part, the perfect surface has already been created. Therefore the first step is complete and the inspection process can continue.

1. On the *Model Tab* select the *Plane* tool.

2. Select the perfect surface opposite datum ***-B-***, drag the cursor orthonormal to this plane, and then release the left mouse button.

3. Enter half of the profile tolerance into the dialog box (it may be negative depending on the direction chosen), which in this case is 5 mm. This is because the feature can vary up to 5 mm in either direction.

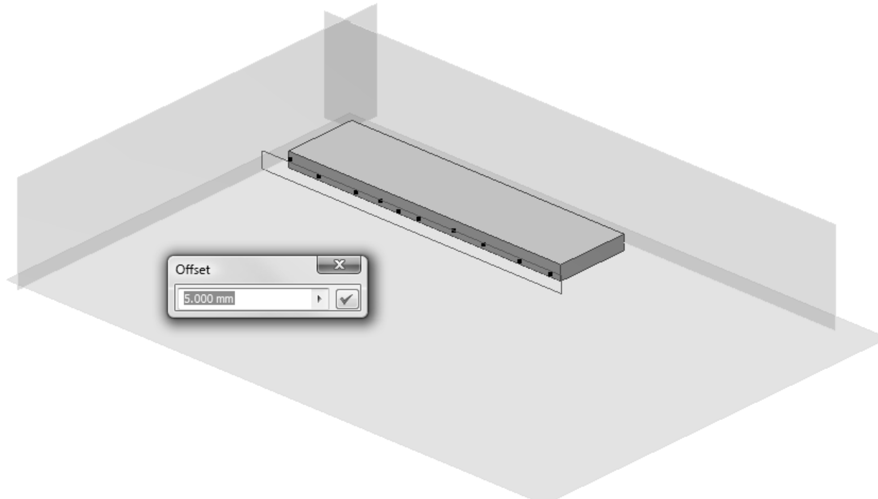


Figure E.20: First tolerance boundary plane.

4. On the *Model Tab* select the *Plane* tool.

5. Select the surface opposite datum ***-B-***, drag in the opposite orthonormal direction this time, and then release the left mouse button.

6. Enter half of the perpendicularity tolerance into the dialog box (it may be negative depending on the direction chosen), which in this case is -5 mm.

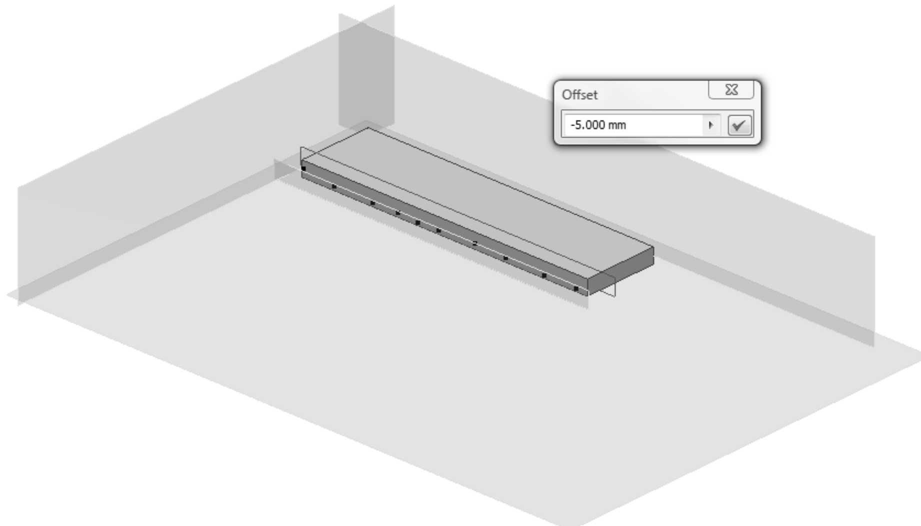


Figure E.21: Second tolerance boundary plane.

7. Using the *Navigation Cube*, orient Block 1 in the top view so the boundary zones appear as straight lines.

8. The last step is to check if all of the inspection points fall within the boundary zone. If they do not, the part does not meet the 10 mm profile tolerance.

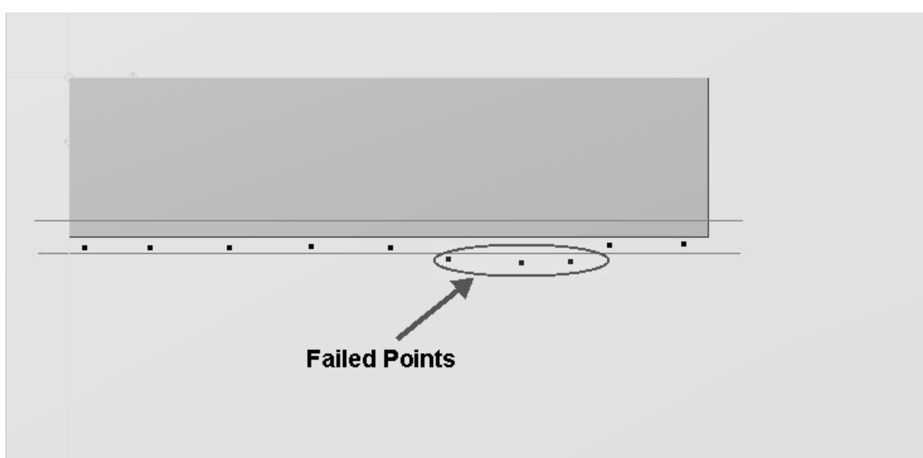


Figure E.22: Top view of perfect plane, tolerance boundary zone, and inspection points.

9. If more detailed information is needed about how far out of tolerance the part is, navigate to the *Inspect Tab* and select the *Distance* tool.

10. Select the failed point of interest and then the tolerance boundary, and a measurement of how out of tolerance the part is can be seen. In this case the point is 1.485 mm out of tolerance.

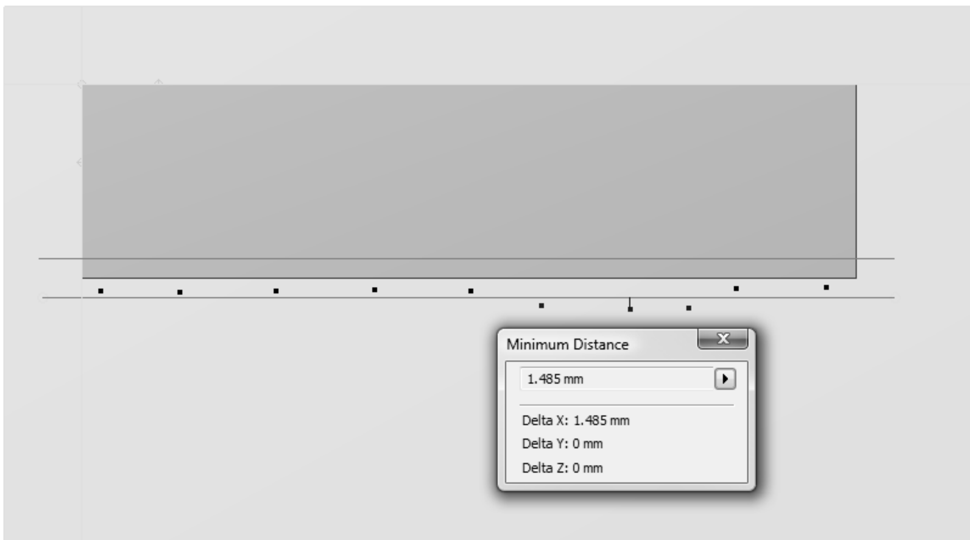


Figure E.23: Top view of failed point.

E.2.5 Orientation Class: *Perpendicularity Tolerance* Fundamentals

The surface opposite datum ***-B-*** is controlled with a geometric tolerance, and the surface which establishes datum ***-B-*** can also be controlled. Note that this is different from the dimensional tolerance, because each surface can be controlled independently. This will be accomplished using an orientation tolerance instead of a location tolerance.

Before, Block 1 was fixtured to the measurement table using the primary, secondary, and tertiary datums specified in the feature control frame. These surfaces served as the datum reference frame for the surface profile geometric tolerance. This was done because the surface profile tolerance needs a baseline coordinate system to measure against.

Datum ***-B-***, however, is part of the coordinate system. A surface that is being controlled cannot reference itself. Additionally datum ***-C-***, which will be established using datum -

***B*-**, cannot be referenced because that would lead to a circular reference. The only datum available to reference is datum **-*A*-**, and in this case the only relationship that can be established between datum **-*B*-** and datum **-*A*-** is an orientation of 90°.

The engineering drawing of the block has been updated to include a *perpendicularity tolerance* and can be seen in Figure E.24. Note: the tolerance is specified in millimeters as a linear value which is equal to the zone width, and is not equal to an angle. This is discussed below.

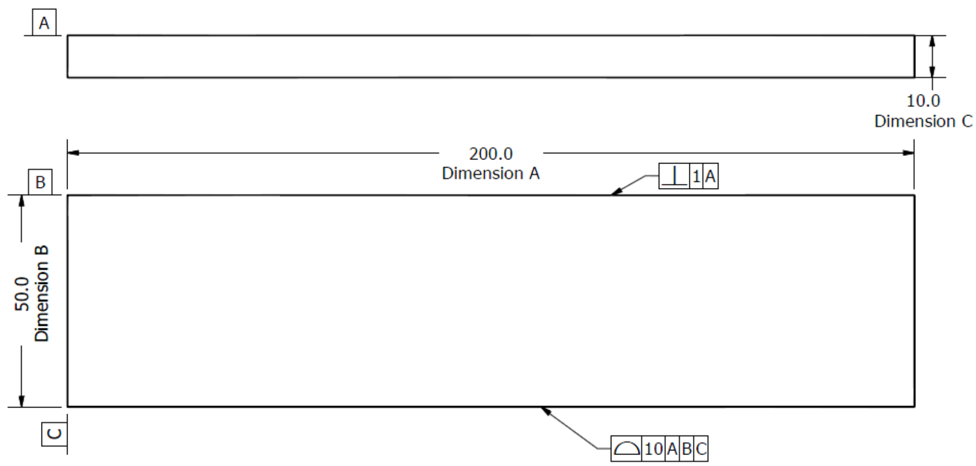


Figure E.24: Engineering drawing of Block 1 with datums and geometric tolerances.

Step 1: Fixture the Block

The only datum available for reference when establishing datum **-*B*-** is datum **-*A*-**, as indicated in Figure E.24. So placing the large flat **-*A*-** datum surface on the table fixtures the block according to the datum reference frame of the perpendicularity tolerance. Only the Z translation, X rotation, and Y rotation have been removed, so the block can still translate in either the X or Y direction and rotate in any orientation about the Z axis. This process can be seen in Figure E.25.

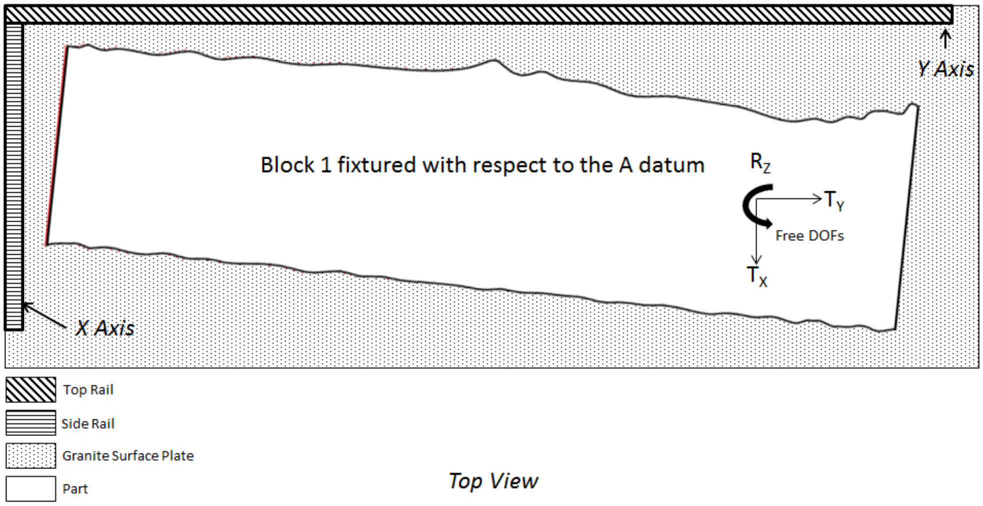


Figure E.25: Block 1 fixtured to the measurement table.

Note: Block 1 can be oriented at any angle with respect to the top measurement rail and the side measurement rail. These orientation angles are not controlled in this case.

Step 2: Create Perfect Surface

A perpendicularity tolerance only has the ability to control orientation and form. There can be no perfect surface established since location is not controlled with additional datum references (remember the three free degrees of freedom). Therefore, only a perfect plane may be established, which is 90° with respect to the ***-A-*** datum. This is specified in the feature control frame, and it can be seen Figure E.26.

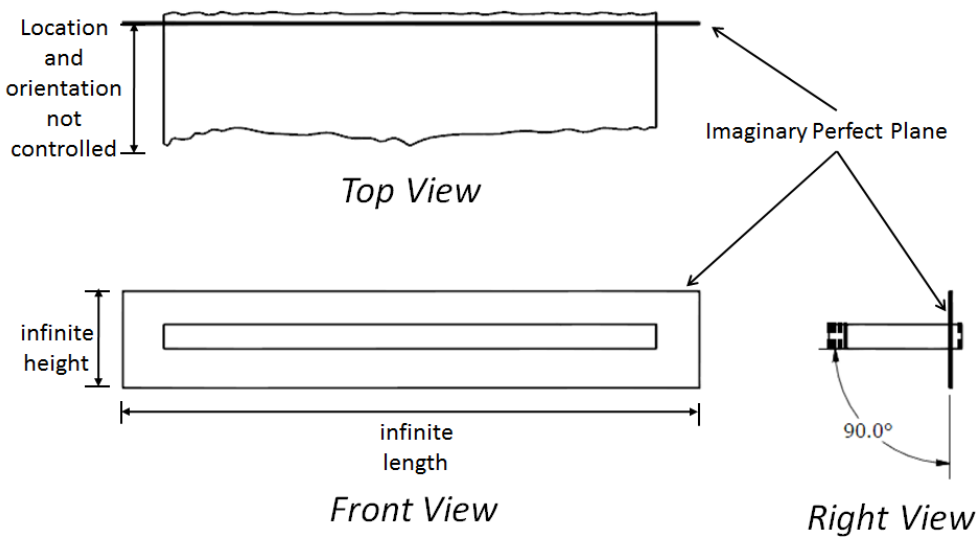


Figure E.26: Noting the orientation of the plane with respect to the **-*A*-** datum.

Step 3: Create the Planar Boundaries

The last step in applying and checking a perpendicularity tolerance is creating the tolerance boundary or zone. Even though the perpendicularity tolerance is an orientation tolerance, the boundary is still specified as a linear zone not an angular tolerance. In this case, the boundary surface consists of two planes, which are 1 mm apart because of the tolerance value, and centered on the perfect plane. The boundary zone is located with respect to the surface it is applied to, and the surface must fit within this boundary. However, the manufactured surface may take any orientation or shape within the zone, as in Figure E.27.

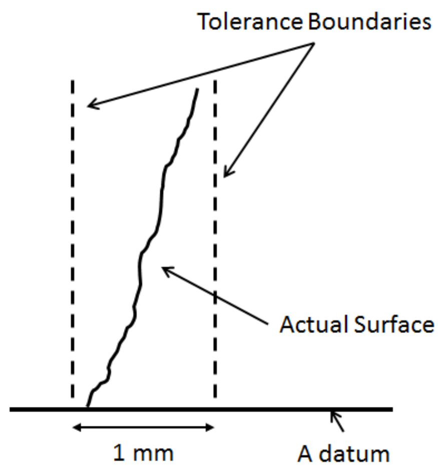


Figure E.27: The planar boundary zones for the perpendicularity tolerance. The boundary zone must be perpendicular to the ***-A-*** datum and must be 1mm apart. The surface may take any orientation or shape within the zone.

This is similar to the case of a profile boundary zone, except no location on the measurement table is specified. Therefore, the block may be oriented anywhere on datum ***-A-*** and the zone must only be perpendicular to datum ***-A-*** due to the single datum reference. Recall, the datum reference frame specifies how many datums are needed to fully constrain the part. This allows the zone to be oriented at any angle with respect to datum ***-B-*** or datum ***-C-***. A few acceptable configurations where the part is fully fixtured can be seen in Figure E.28.

Note: While it is true the block may be oriented anywhere on datum ***-A-***, it is usually best to use the additional measurement planes to help secure the part during inspection so it does not move between measures.

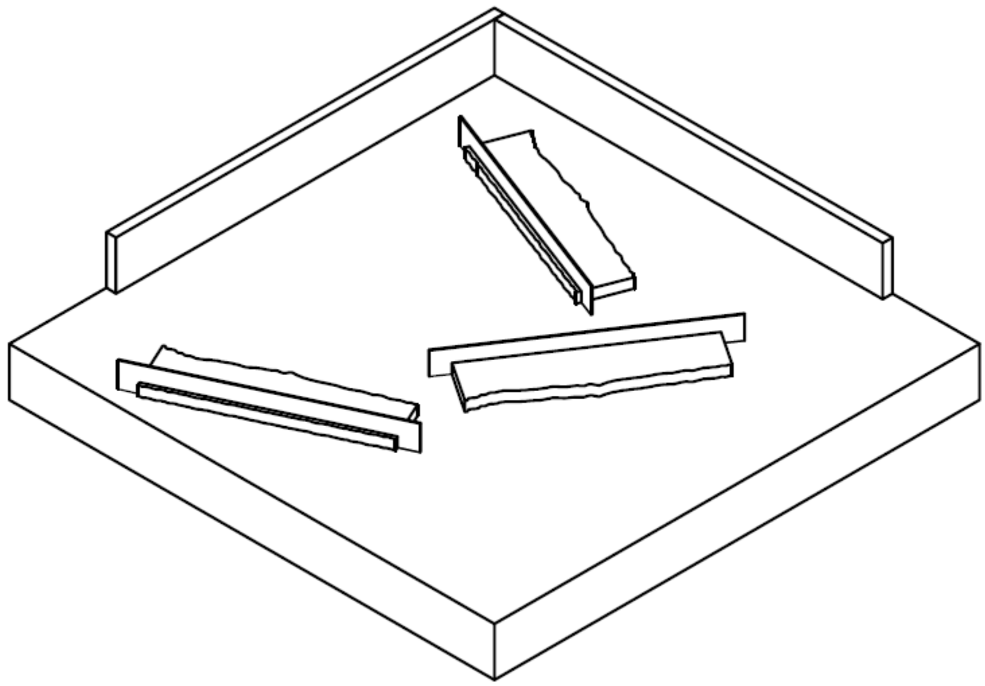


Figure E.28: Creating the planar boundary zones for the perpendicularity tolerance. The boundary zone must only be perpendicular to the ***-A-*** datum and may take any orientation with respect to datum ***-B-*** and datum ***-C-***.

Perpendicularity Special Cases: Multiple Datum References

If the perpendicularity tolerance was applied to the surface opposite the ***-C-*** datum, as seen in Figure E.29, more angular control would be applied to the feature of interest.

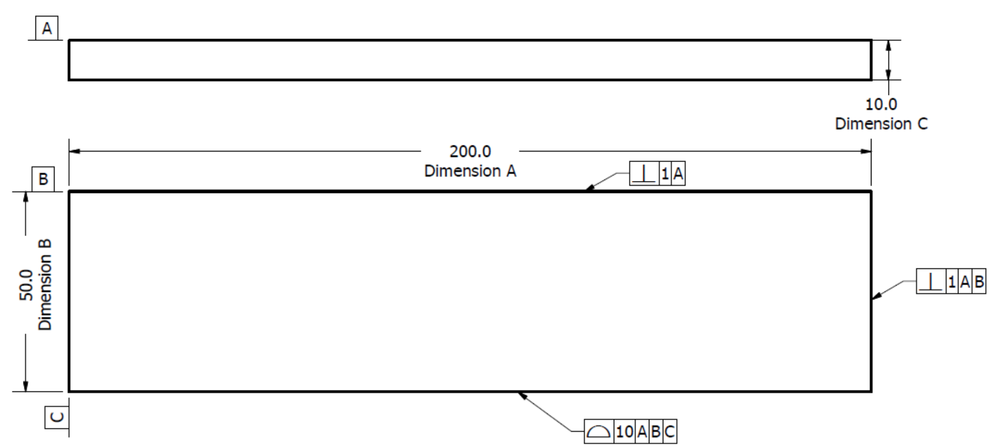


Figure E.29: Perpendicularity tolerance added to surface opposite of ***-C-***.

In this case, Block 1 would be fixtured with Datum ***-A-*** and Datum ***-B-***, as in Figure E.30.

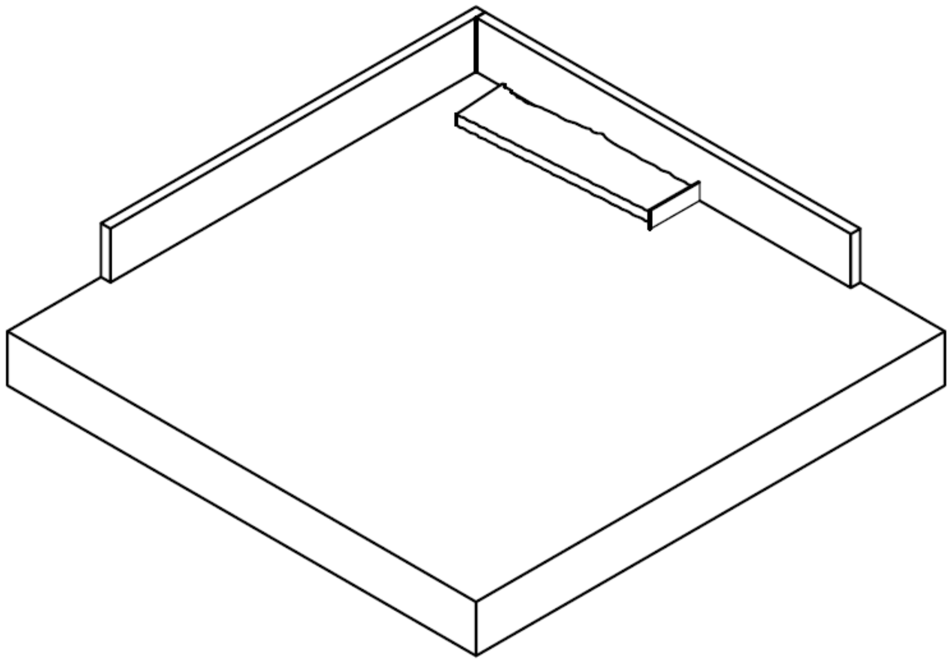
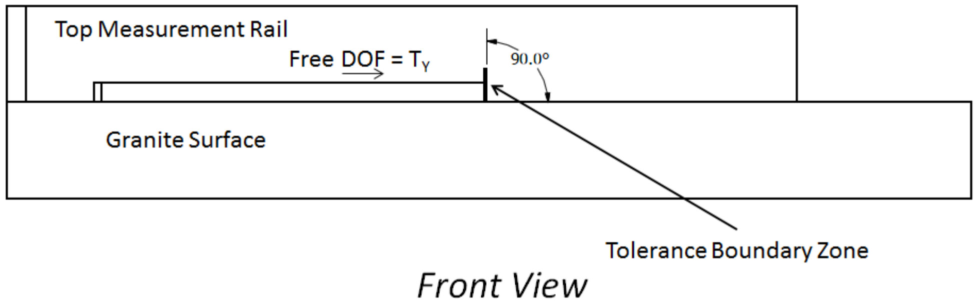
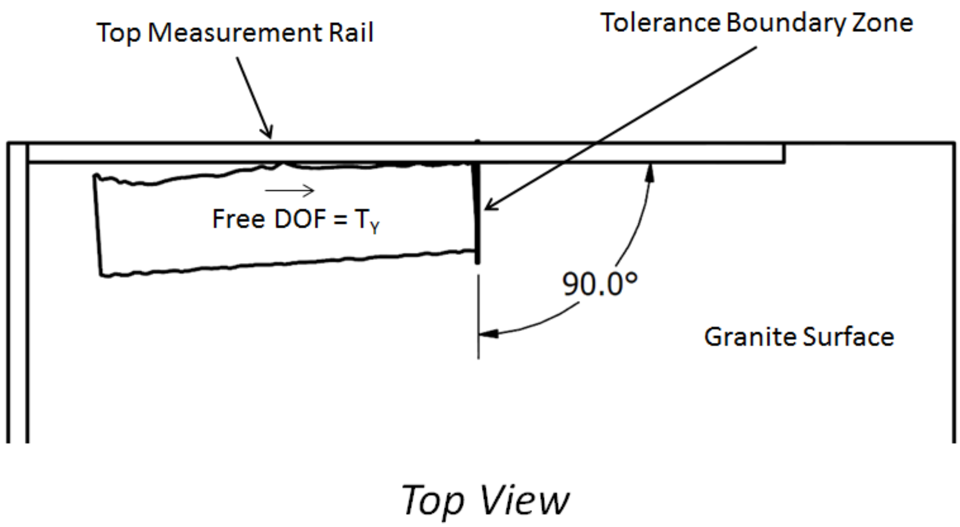


Figure E.30: Block 1 fixtured with respect to the ***-A-*** datum and ***-B-*** datum.

The perfect plane for this tolerance would then be required to be 90° to both datum ***-A-***, see Figure E.31(a) and datum ***-B-***, see Figure E.31(b). In this case, the tolerance controls the orientation of the surface in two directions.



(a) The planar boundary zone 90° with respect to datum ***-A-***.



(b) The planar boundary zone is also 90° with respect to datum ***-B-***.

Figure E.31: Boundary zones for perpendicularity tolerance with respect to two datums.

E.2.6 Perpendicularity Tolerance Inspection Example

The first step in inspecting a perpendicularity tolerance applied to a manufactured block is to take inspection points off of the surface of interest.

Establishing Inspection Points

1. Begin by starting Autodesk Inventor and opening the `Block_1_CMM_Measurement_Perpendicularity.ipt`.

2. Initialize and Calibrate the CMM according to the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* found in Appendix B.

3. Fixture the block to the CMM measurement table using the primary datum designated in the engineering drawing and the feature control frame. To ensure stability, place datum -***C***- on the top measurement rail and the surface opposite of datum -***B***- on the side measurement rail.

Figure E.32: Fixturing Block 1 to inspect the perpendicularity tolerance on datum -***B***-.

4. Take 10 CMM measurements along datum -***B***- according to the *Instruction Set: Taking Surface Inspection Points* found in Appendix C. Be sure to keep the CMM base and the part still during the entire measurement process.

5. After all the points are taken, the Autodesk Inventor screen should look similar to Figure E.33. If all the points are not visible, navigate to the *View Tab* and select the *Wireframe Shading*.

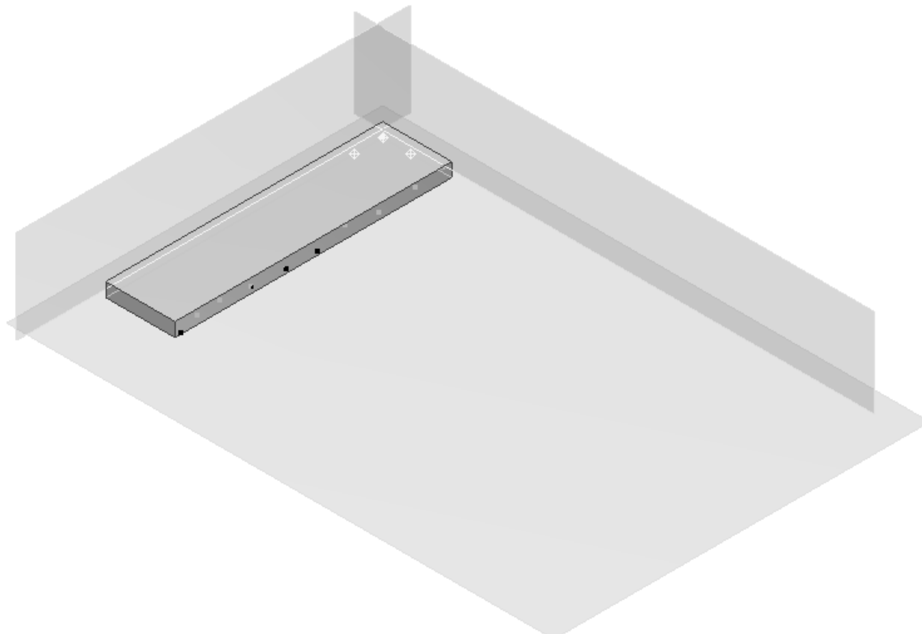


Figure E.33: CMM inspection points in Autodesk Inventor.

Establishing the Perfect Plane

According to the perpendicularity tolerance, the only requirement for the perfect surface is that it is 90° to datum ***-A-***.

1. Use the *Navigation Cube* and orient the part in the top view

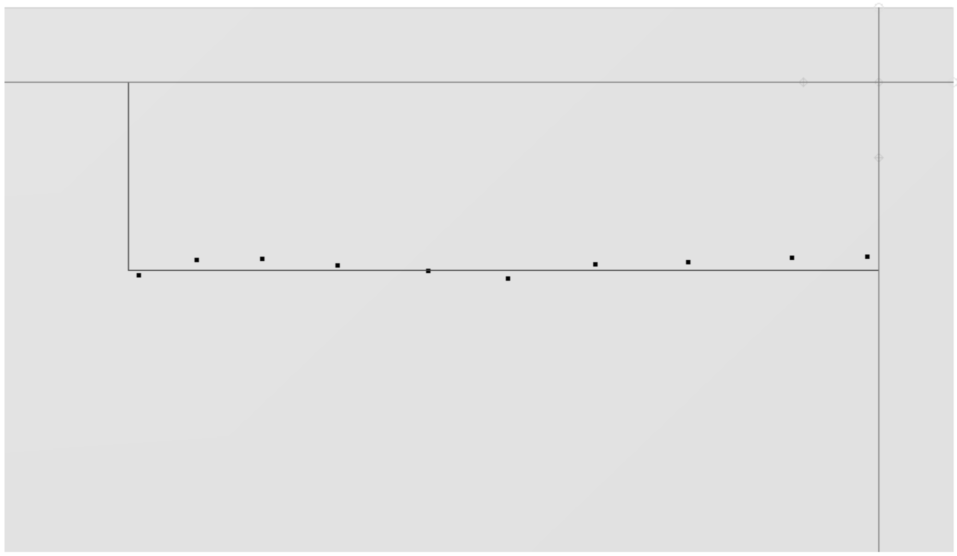


Figure E.34: Top view of CMM inspection points.

2. On the *Model Tab* select the *Create 2D Sketch* tool.

3. Select the XY Plane

4. On the *Sketch* tab select the *Project Geometry* tool and select on the first and last inspection point. This is done so a rough estimate of a best fit line can be created. If the points cannot be selected, place the mouse cursor over the point, right click, on the context menu choose *Select Other*, use the arrows to highlight the point, and then click the middle green button to select the point.

5. On the *Sketch* tab select the *Line* tool and select the first and last inspection points.

6. Create a line between the first and last projected points.

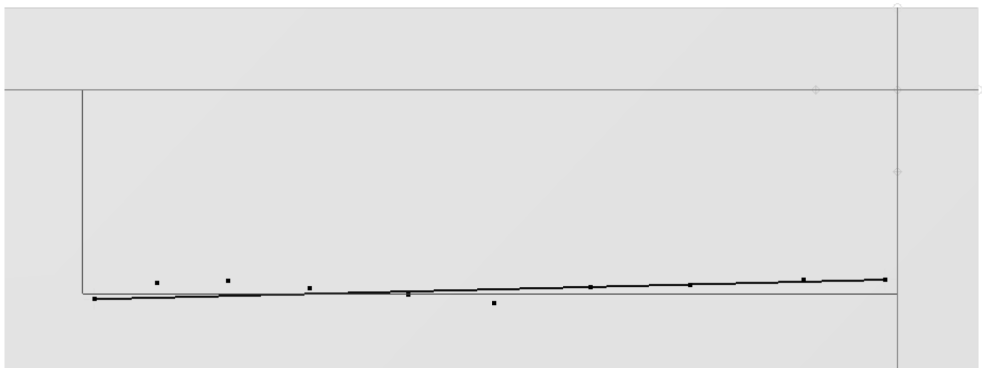


Figure E.35: Best fit line created between the first and last projected points.

7. On the *Model Tab* select the *Plane* tool.

8. Hold down the Ctrl button and select the previously created best fit line and the XY plane.

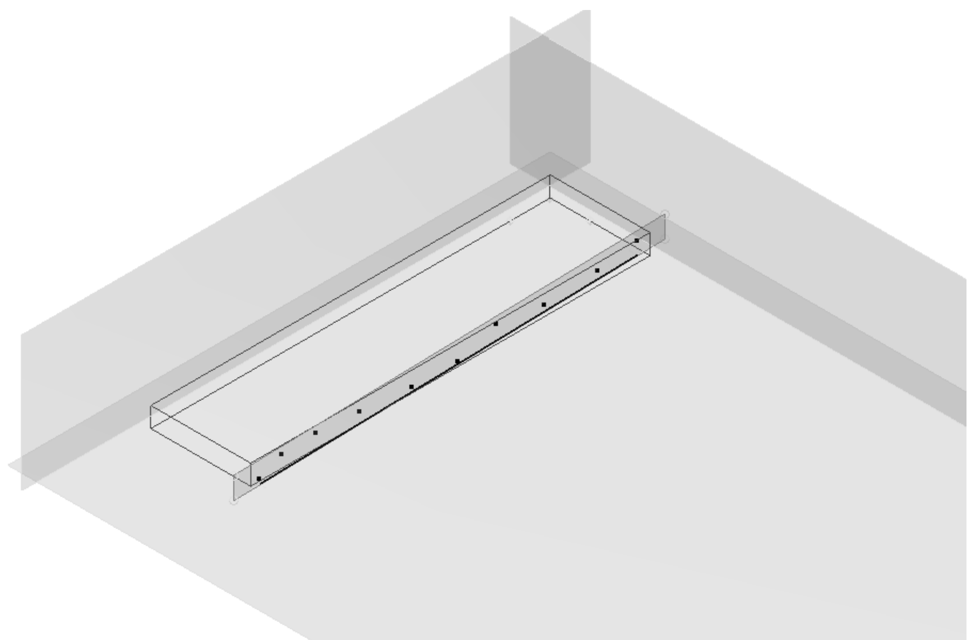


Figure E.36: Creating the perfect plane from the best fit axis and the base XY plane.

9. Enter 90° into the Angle prompt and select the check mark.

10. On the *Model Tab* select the *Plane* tool.

11. Select the newly created plane and drag to the right.

12. Enter 20 mm in the offset box. This will create an offset plane to measure all the inspection points against. By offsetting it 20 mm, all points will occur on the same side of the plane and all the direction vectors from the point to the plane will match.

13. Right click on the last work plane created in the family tree and select Expand All Children.

14. Right mouse click on all children of the newly created plane except for the XY plane and uncheck the visibility. Also uncheck the visibility for the sketched line.

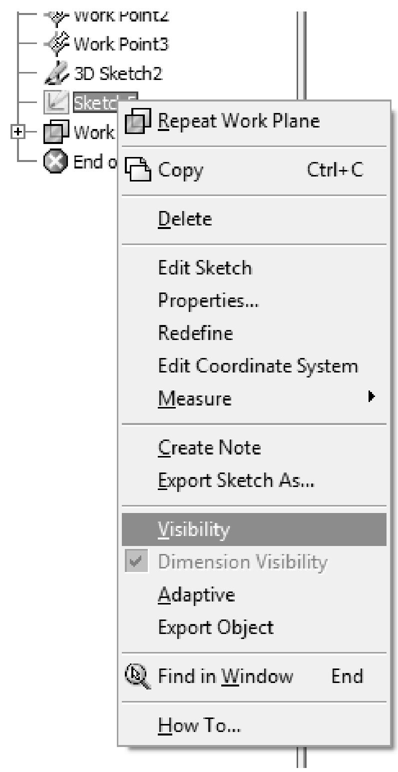


Figure E.37: Expanding the perfect plane feature.

15. Use the *Navigation Cube* and orient to the top view of the part.

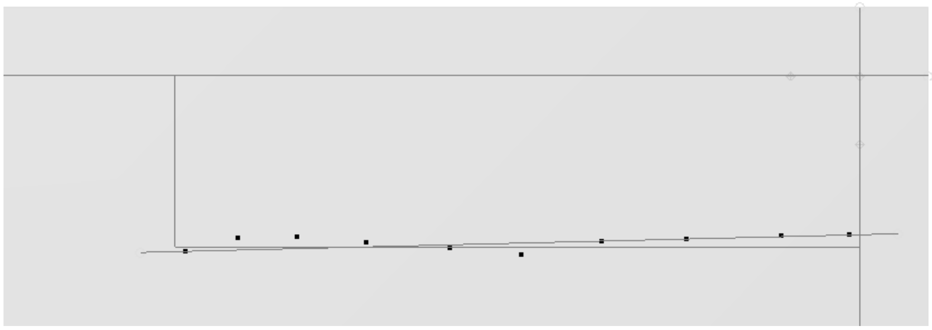


Figure E.38: Top view of the perfect plane and the CMM points.

Measure the Variation

The final step in inspecting the perpendicularity tolerance is measuring the maximum variation between the two most extreme inspection points. This maximum variation needs to be less than the tolerance value, as seen in Figure E.39.

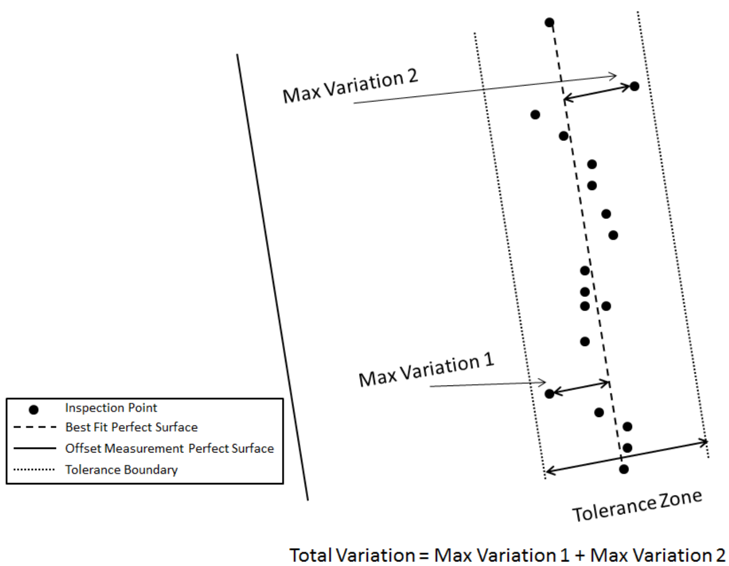


Figure E.39: Finding the maximum variation from the perfect plane.

Unfortunately because perpendicularity tolerances are typically tight, it is very difficult to visually examine the inspection points to see which side of the plane they occur on. This makes it very difficult to accurately calculate the maximum variation. To avoid this problem, a 20 mm offset plane is created so all the inspection points occur on the same side of the measurement plane. By doing this, it eliminates the need to track the magnitude of the distance measures and makes the inspection process simpler than if the simulated perfect surface is used. Once the inspection is complete, the distance between the two most extreme points will then be calculated in Microsoft Excel. This value will be the maximum variation and its value must be less than or equal to the tolerance value. A schematic of this process can be seen in Figure E.40.

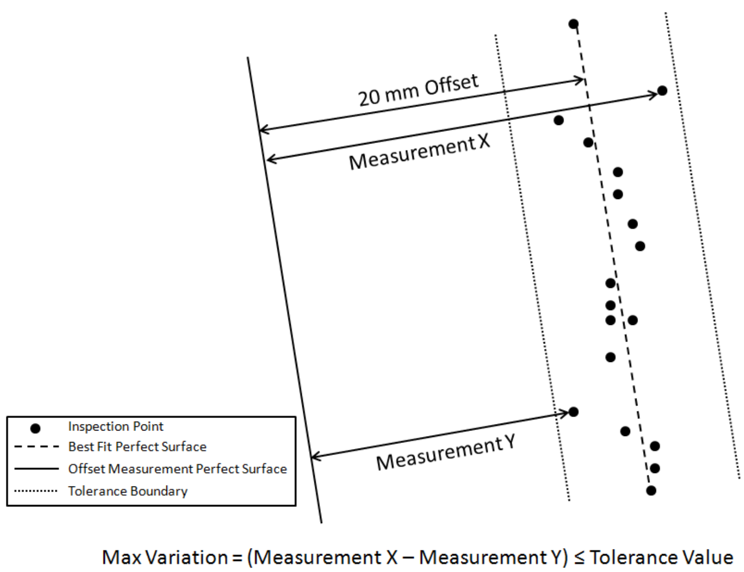


Figure E.40: Schematic of the perpendicularity inspection process with an offset plane.

1. Open the perpendicularity_insepction.xlsx spreadsheet
2. Enter the perpendicularity tolerance value in the corresponding cell (a yellow cell located under the measurement data rows), in this case 1 mm.
3. Switch to the Autodesk Inventor window
4. On the *Inspect Tab* select the *Distance* tool.
5. Select the first measure point and the offset plane.

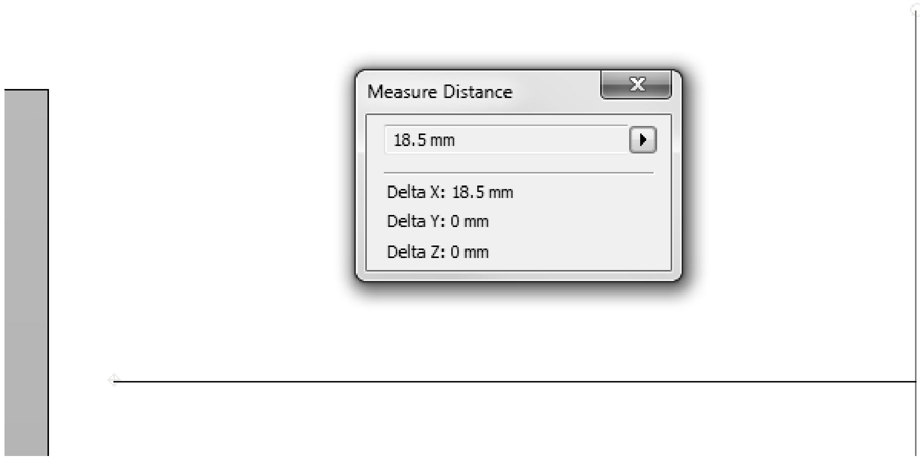


Figure E.41: Distance between the inspection point and the perfect plane.

6. Switch to the Microsoft Excel window
7. Enter the first measurement distance, which in this case is 18.5 mm. This distance entered will always be the absolute value of the measure distance and therefore always positive.
8. Repeat this procedure for the remaining 9 points.
9. Examine the max variation and whether it is less than the perpendicularity tolerance.

User Input				
Point	Absolute Value of Measure (must be positive)	*Point 1 - Points*	*Point 2 - Points*	*P*
1	18.5	*0*	*1.444*	
2	19.944	*-1.444*	*0*	
3	20.265	*-1.765*	*-0.321*	
4	20.009	*-1.509*	*-0.065*	
5	20.095	*-1.595*	*-0.151*	
6	20.135	*-1.635*	*-0.191*	
7	20.223	*-1.723*	*-0.279*	
8	20.315	*-1.815*	*-0.371*	
9	20.039	*-1.539*	*-0.095*	
10	20	*-1.5*	*-0.056*	
Perpendicularity Value	1			
Max Variation	*1.815*			
Pass/Fail?	FAIL			

Figure E.42: Calculated maximum variation is 1.815,greater than the 1 mm limit.

E.2.7 Form Class: *Surface Flatness Tolerance* Fundamentals

The next category of tolerance that will be discussed is the form tolerance, which controls the form of an object. The specific tolerance that will be applied to the drawing of Block 1 is a flatness tolerance on datum ***-A-***. This can be seen in Figure E.43.

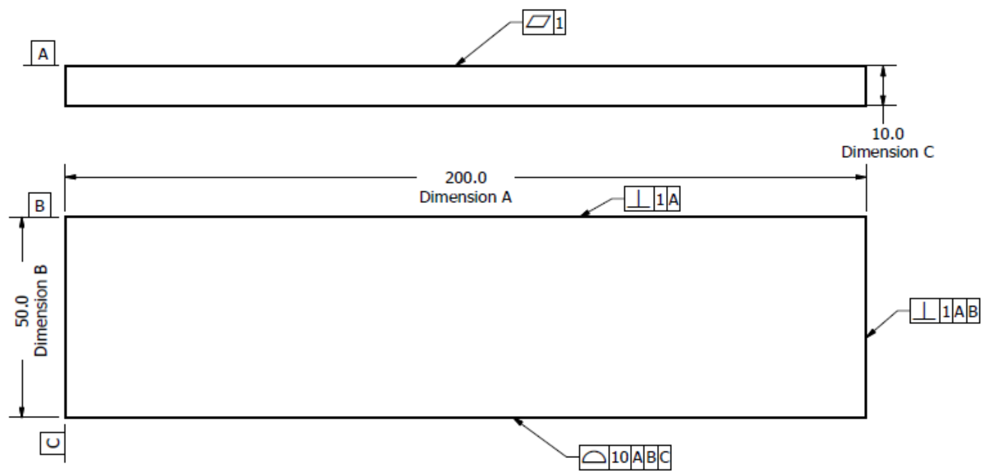


Figure E.43: Block 1 with a flatness tolerance applied.

A flatness tolerance may be applied to any surface, but no datum reference is needed in the feature control frame. This is because the flatness of a feature is measured relative to itself. Due to this property, form tolerances are typically applied to the feature used as the primary datum (just as in Figure E.43), since no baselines are needed for measurement.

Step 1: Fixture the Block

Since there are no datum references in the feature control frame, the block may be measured in any orientation in free space. However, it is typical to fixture the block down in some orientation on a measurement table to insure accurate, precise, and repeatable measurement conditions.

Step 2: Create Perfect Surface

In the case of a flatness tolerance, the perfect surface is a plane. This plane can be located anywhere and in any orientation. Any combination of three inspection points can form the basis for this plane.

Step 3: Create the Planar Boundaries

The planar boundaries, as in the previous two tolerance classes, are two planes 1 mm apart and are centered on the perfect plane. The surface may take any shape within this planar boundary. An example of the planar boundary created can be seen in Figure E.44. The part is configured randomly to demonstrate how orientation and location are not specified by the flatness tolerance.

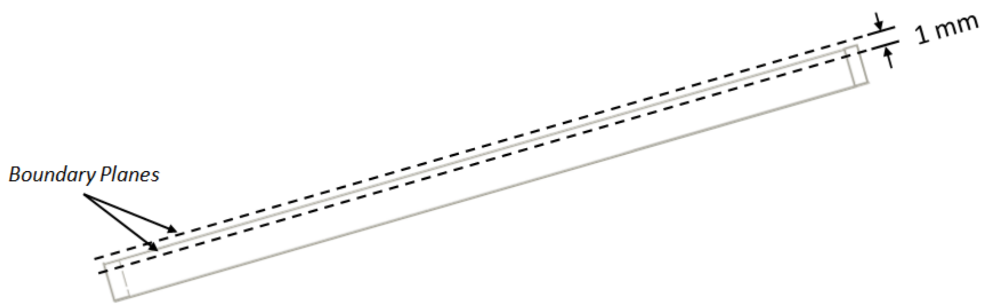


Figure E.44: Flatness boundaries for Block 1.

E.2.8 Surface Flatness Tolerance Inspection Example

Inspection of the surface flatness tolerance is similar to the inspection of the perpendicularity tolerance. The difference is the measurement plane is formed from only three inspection points and no other reference features are needed. This plane may then be used to measure the variation distance between the extreme points.

Establish Inspection Points

The first step in inspecting a flatness tolerance applied to a manufactured block, is to take inspection points off of the surface of interest.

Establishing Inspection Points

1. Begin by starting Autodesk Inventor and opening the `Block_1_CMM_Measurement_Profile.ipt`.

2. Initialize and Calibrate the CMM according to the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* found in Appendix B.

3. There is no required fixturing of the block according to the datum reference frame, but to ensure stability, place the surface opposite of datum **-*A*-** on the granite plane, place datum **-*C*-** on the top measurement rail, and place datum **-*B*-** on the side measurement rail.

Figure E.45: Fixturing Block 1 to inspect the flatness tolerance on datum **-*A*-**.

4. Take 10 inspection points along datum ***-A-*** according to the *Instruction Set: Taking Surface Inspection Points* found in Appendix C. Be sure to keep the CMM base and the part still during the entire measurement process.

5. After all the points are taken, the Autodesk Inventor screen should look similar to Figure E.46. If all the points are not visible, navigate to the *View Tab* and select the *Wireframe Shading*.

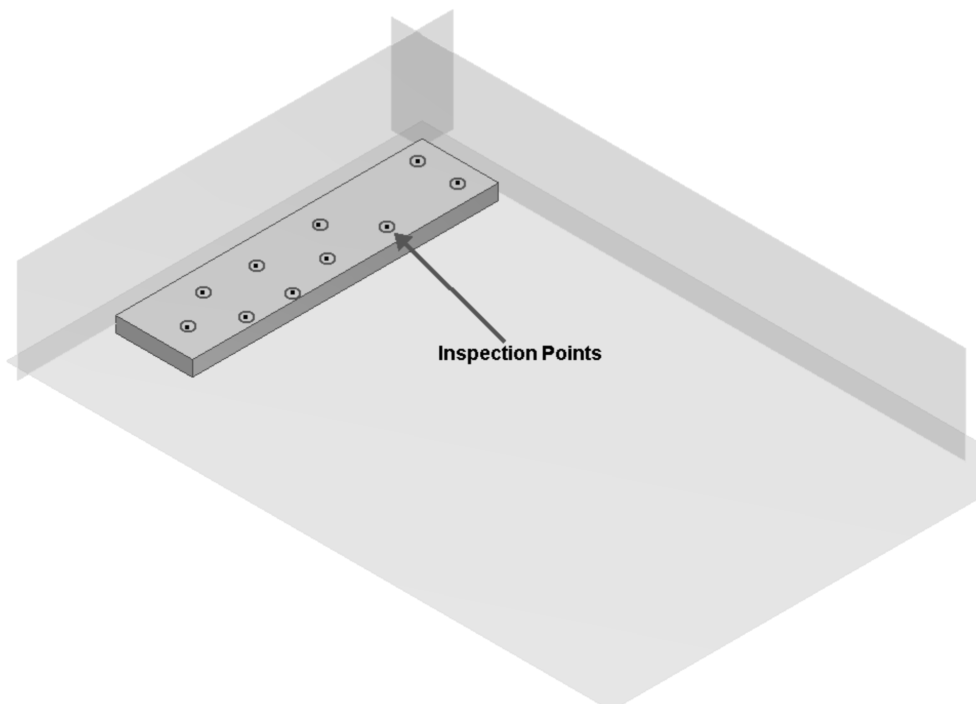


Figure E.46: CMM inspection points in Autodesk Inventor.

Establish the Perfect Plane

According to the flatness tolerance, the only requirement for the perfect surface is that it is a plane located anywhere created from three inspection points.

1. Use the *Navigation Cube* and orient the part in the top view.

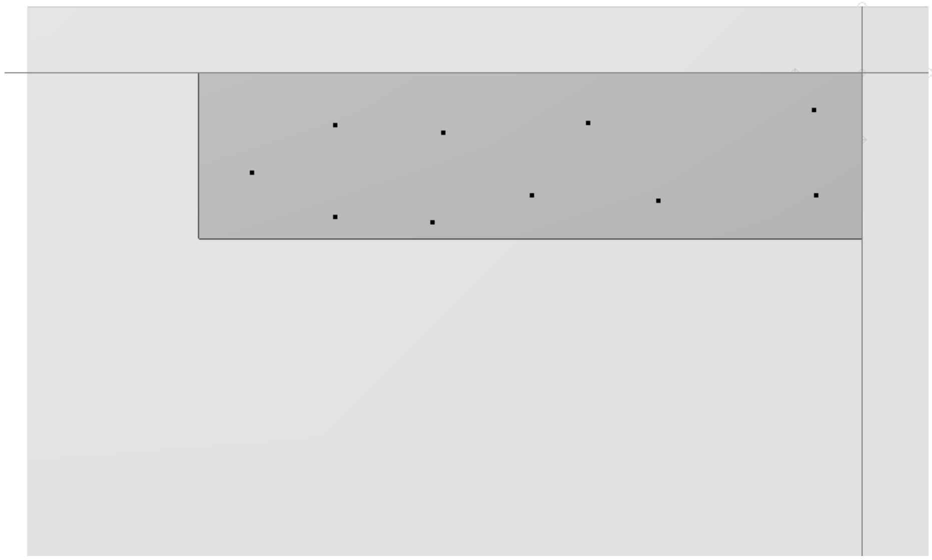


Figure E.47: Top view of CMM inspection points.

2. On the *Model Tab* select the *Plane* tool.

3. Hold down the Ctrl button and select three evenly spaced points to create a plane.

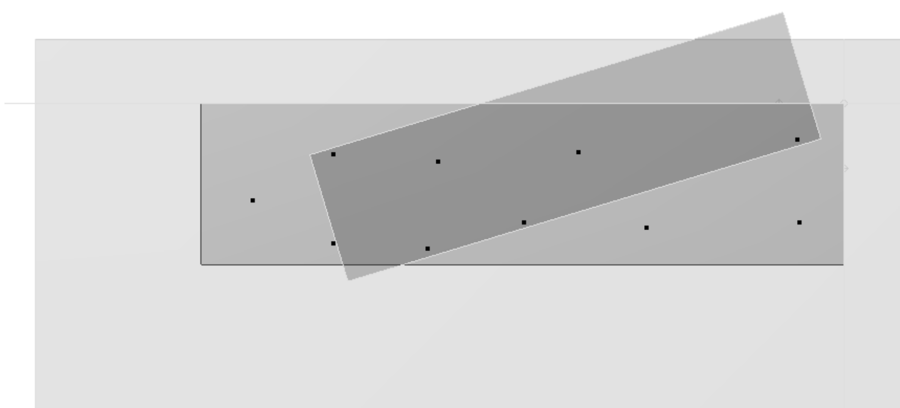


Figure E.48: Top view of CMM inspection points and perfect plane.

4. Since the inspection points will be very close to this perfect plane, a new offset plane will be created in the same orientation. This will serve as a reference plane for measurement and is similar to the perpendicularity inspection example.

5. On the *Model Tab* select the *Plane* tool.

6. Select the previously created plane and drag orthonormal to this plane.

7. Enter 20 mm for the offset value.

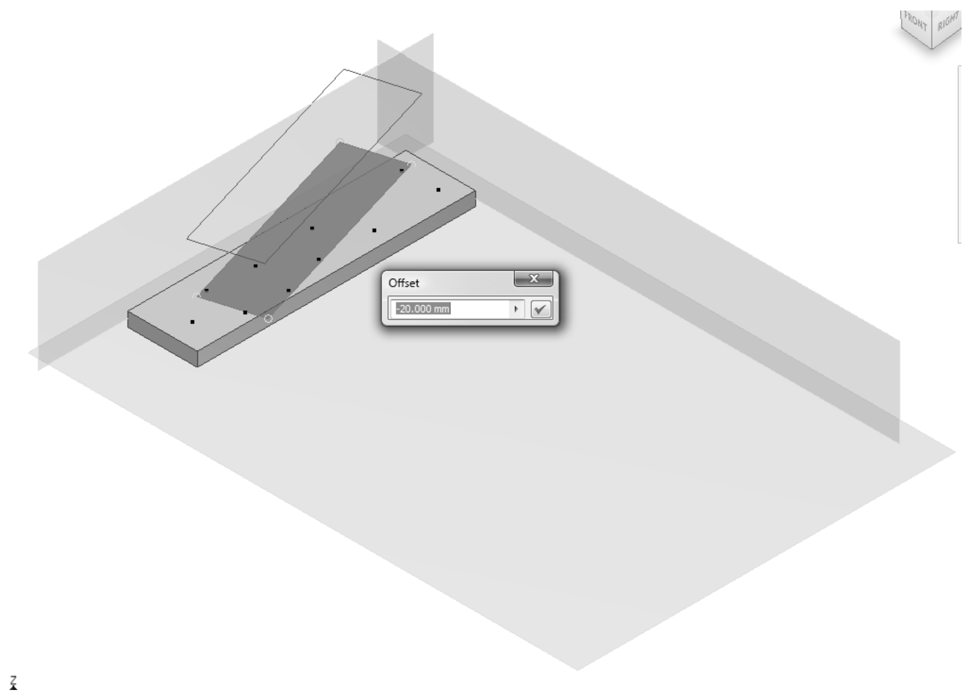


Figure E.49: Isometric view of the Inspection block and the two planes created.

8. In the *Model Tree*, select the best fit plane, right mouse click, and uncheck the visibility.

9. Use the *Navigation Cube* to orient the part so the shifted best fit plane appears as close to a line as possible, as seen in Figure E.50.

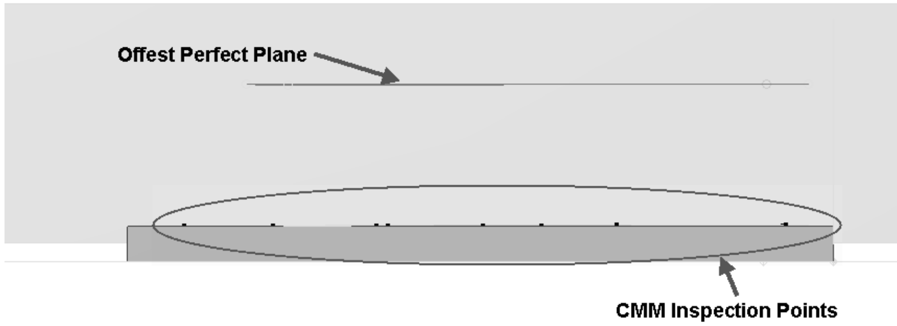


Figure E.50: Side view of the block, inspection points, and the offset perfect plane.

Measuring the Variation

The final step in inspecting the flatness tolerance is measuring the distance from each point to the shifted best fit. The distance between the two most extreme points will then be calculated, and this value will be the maximum variation. This value must be less than or equal to the tolerance value. At this point, the process is equivalent to the perpendicularity inspection process.

1. Open the flatness_insepction.xlsx spreadsheet.

2. Enter the flatness tolerance value in the corresponding window.

3. Switch to the Autodesk Inventor window.

4. On the *Inspect Tab* select the *Distance* tool.

5. Select the first measure point and then the perfect plane, as seen in Figure E.51.

Figure E.51: Distance between the inspection point and the perfect plane.

6. Switch to the Microsoft Excel Window.

7. Enter the first measurement distance, which is 19.857 mm.

8. Repeat this procedure for the remaining 9 points.

9. Examine the max variation and whether it passes the flatness tolerance.

	A	B	C	D
1	User Input			
2	Point	Absolute Value of Measure (must be positive)	*Point 1 - Points*	*Point 2 - Points*
3	1	19.857	*0*	*0.143*
4	2	20	*-0.143*	*0*
5	3	19.978	*-0.121*	*0.022*
6	4	20.207	*-0.35*	*-0.207*
7	5	20	*-0.143*	*0*
8	6	19.974	*-0.117*	*0.026*
9	7	20.022	*-0.165*	*-0.022*
10	8	20.192	*-0.335*	*-0.192*
11	9	20.141	*-0.284*	*-0.141*
12	10	20	*-0.143*	*0*
13	Flatness value	1		
14				
15	*Max Variation*	*0.35*		
16	Pass/Fail?	PASS		

Figure E.52: Calculated maximum variation is 0.35 which is less than the limit of 1 mm.

E.2.9 Position Class: *True Position* Tolerance Fundamentals

The final tolerance class which will be considered is the position tolerance. This tolerance can be applied to a feature of size, which is any feature which can be grabbed with calipers.

Examples include a pair of surfaces, a hole, or an extruded cylinder. On this feature of size, a perfect feature and tolerance boundary zone can be established. During this laboratory the application of a true position tolerance to a cylinder will be considered, but many of the same concepts also hold when applied to a hole. Block 1 has been altered to incorporate a cylinder, and an updated drawing can be seen in Figure E.53.

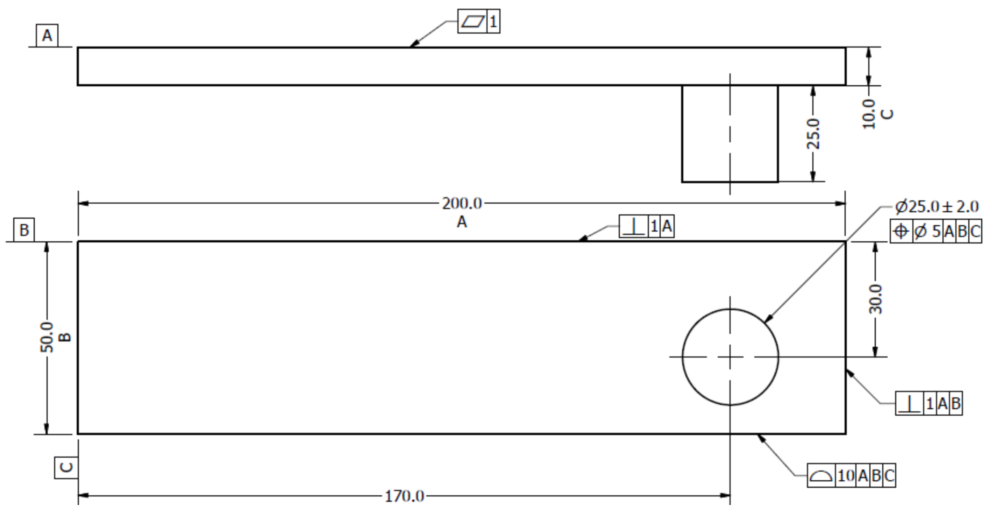


Figure E.53: Cylinder on Block 1.

Items to note about the true position tolerance versus previous geometric tolerances:

- The true position tolerance is added to a feature that also incorporates a dimensional size tolerance. This is because a true position tolerance can only control a feature of size.

- In this case there is a diametral symbol, ∅, before the tolerance value. This specifies the tolerance zone should be circular, not rectangular as in previous geometric tolerances.

- The dimension that the true position tolerance is added to retains its dimensional size tolerance. In this case, the tolerance is applied regardless of the feature size, but in other cases material modifiers may be added in the geometric tolerance. These modifiers allow the dimensional tolerance and the geometric tolerance to work in unison to control the feature. This is beyond the scope of this laboratory.

Step 1: Fixture the Block

Block 1 is again fixtured into the measurement table using the primary, secondary, and tertiary datums specified in the feature control frame. This fixturing can be seen in Figure E.54.

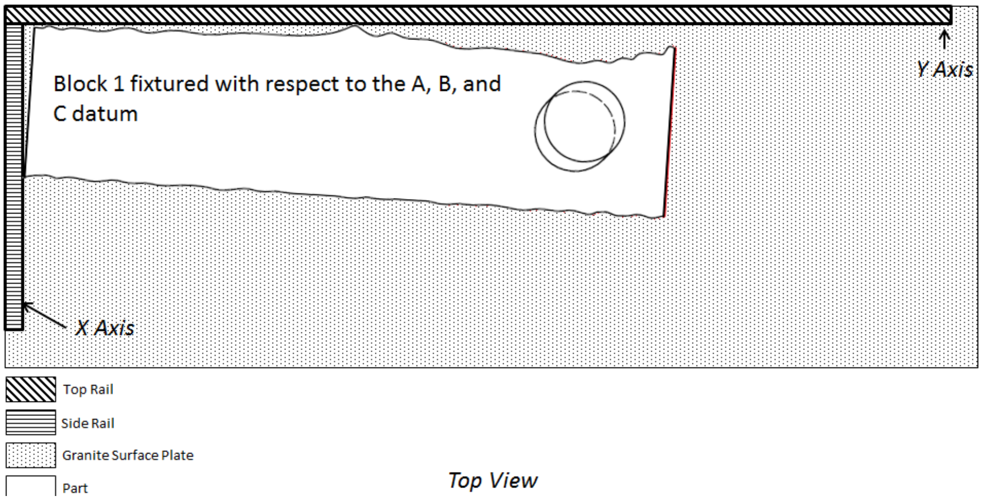


Figure E.54: Block 1 fully fixture to the measurement table.

Step 2: Create the Perfect Feature

This true position tolerance is added *Regardless of Feature Size*, and true position only controls the position and orientation of the cylinder. The feature on the cylinder that defines the position and orientation is the axis, so the perfect feature is a virtual axis. According to Figure E.53, it is located 170.0 mm from datum ***-C-*** and 30 mm from datum ***-B-***. The axis is also 90° with respect to the ***-A-***, ***-B-***, and ***-C-*** datum and is 25.0 mm high. The perfect axis can be seen in Figure E.55.

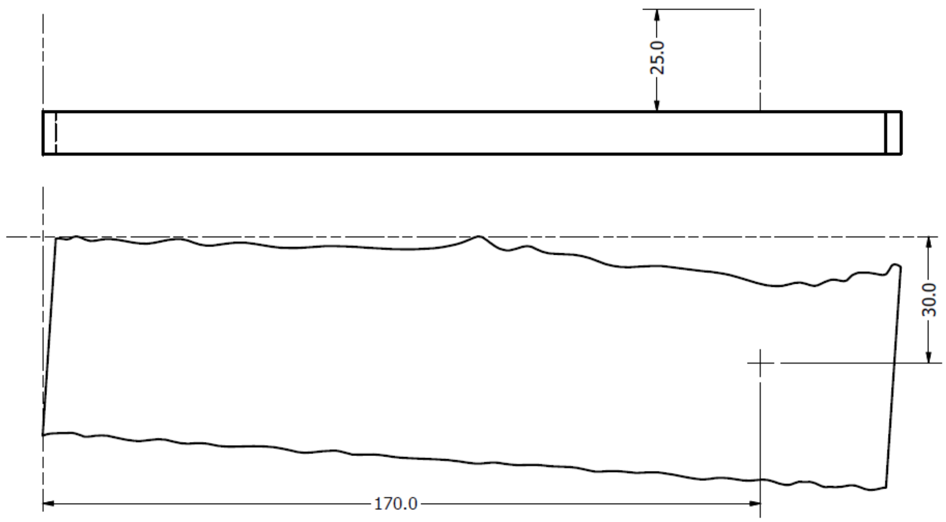


Figure E.55: The perfect feature is a virtual axis.

Step 3: Creating the Tolerance Boundary

Once the virtual, perfect axis is created, the tolerance boundary can be established. According to the feature control frame, the tolerance boundary has a value of 5 mm. Instead of a planar boundary, as in the previous tolerance classes, this boundary is a cylinder which is 5 mm in diameter by 25.0 mm tall. This is due to the diametral symbol, ∅, before the tolerance value. This cylinder is centered on the perfect axis and can be seen in Figure E.56. The actual axis for the cylinder must then fall within the cylindrical boundary zone for the feature to satisfy the requirements of the true position geometric tolerance.

Note: If the diametral indicator was not present, the tolerance zone would revert to 5 mm by 5 mm by 25.0 mm cube.

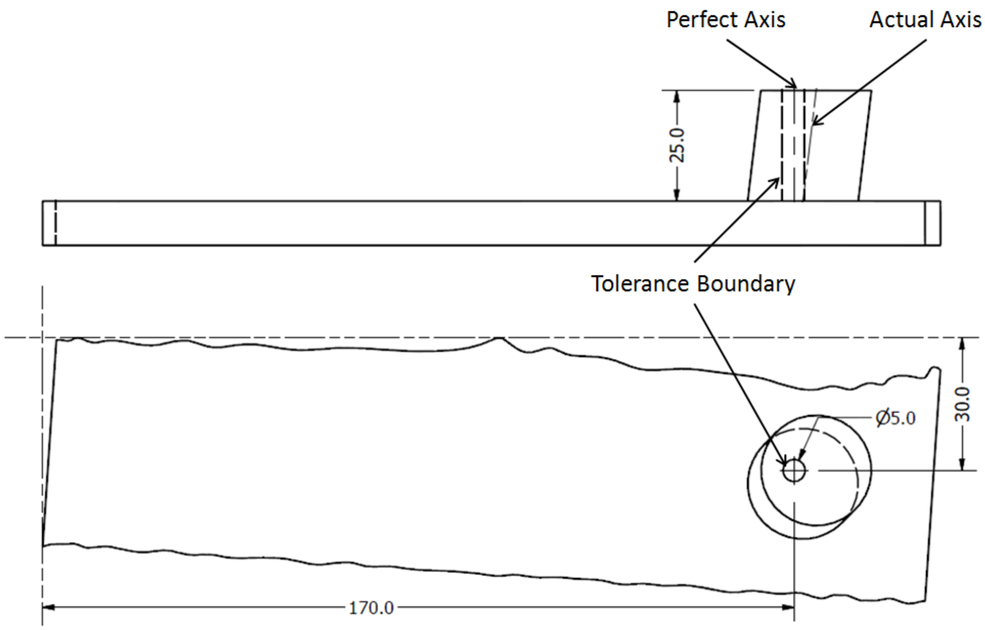


Figure E.56: The perfect feature is a virtual axis.

Position Tolerance Special Cases: Feature Modifiers

In the previous case, the size of the cylinder is only controlled by the dimensional tolerance, while the orientation and position are controlled by the true position tolerance. This is because the tolerance is applied *Regardless of Feature Size*. However, if the tolerance is applied at *Maximum Material Condition (MMC)* or *Least Material Condition (LMC)*, the true position tolerance would also be controlled by the dimensional size tolerance. These modifiers allow the tolerance boundary zone to expand or contract depending on the size feature. These concepts are not discussed in this laboratory.

E.2.10 Surface Flatness Tolerance Inspection Example

Inspection of the true position tolerance is different from the inspection of the previous tolerances discussed, because in this case the true position tolerance is applied to a cylinder instead of a surface. As a result, a different function in Autodesk Inventor must be used to measure the position of the cylinder.

Establishing Inspection Points

1. Begin by starting Autodesk Inventor and opening the `Block_1_CMM_Measurement_True_Position.ipt.`

2. Initialize and Calibrate the CMM according to the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* found in Appendix B.

3. Fixture the block to the CMM measurement table using the primary, secondary, and tertiary datums designated in the feature control frame found on engineering drawing. This can be seen in Figure E.57.

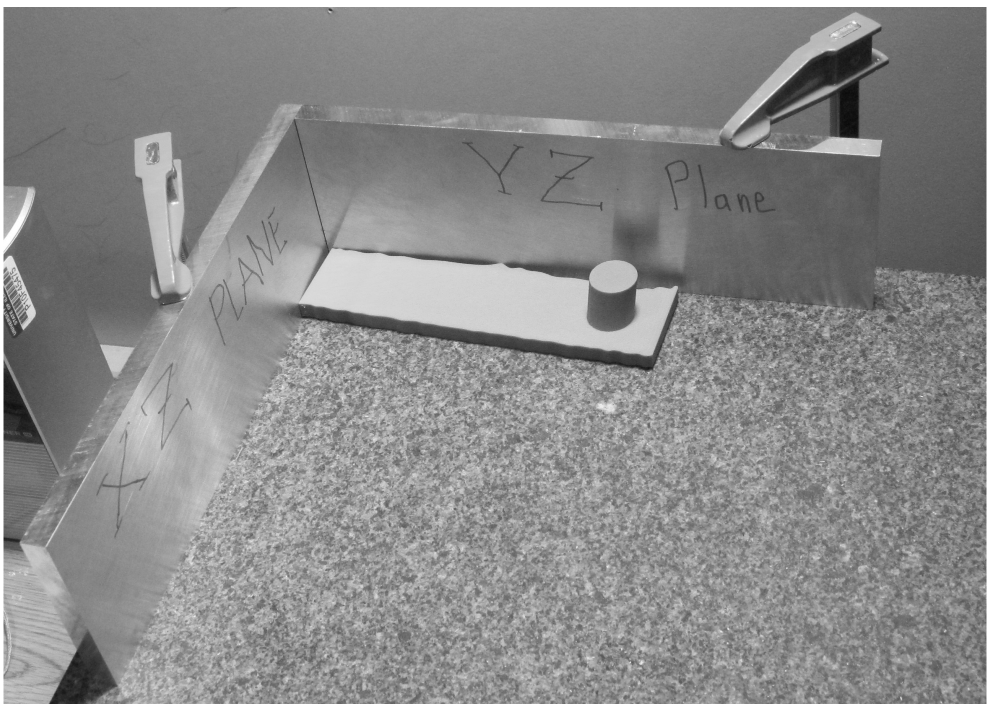


Figure E.57: Fixturing Block 1 to inspect the true position tolerance.

4. Take all the necessary CMM measurements of the cylinder to the *Instruction Set: True Position Inspection Points* found in Appendix D. Be sure to keep the Microscribe base and the part still during the entire measurement process.

Measure the Inspection Circles

The next step in the inspection process is to measure the size of the inspection circles. This is done using the *Distance* tool.

1. On the *Inspect Tab* select the *Distance* tool.

2. Select the first measurement circle and note its diameter as in Figure E.58. (To select the circle you may have to *Right Click* ⇒ *Select Other* and then cycle through until the circle is highlighted. Then click the green middle button.) In this case the diameter is 24.197 mm, which falls within the dimensional size tolerance.

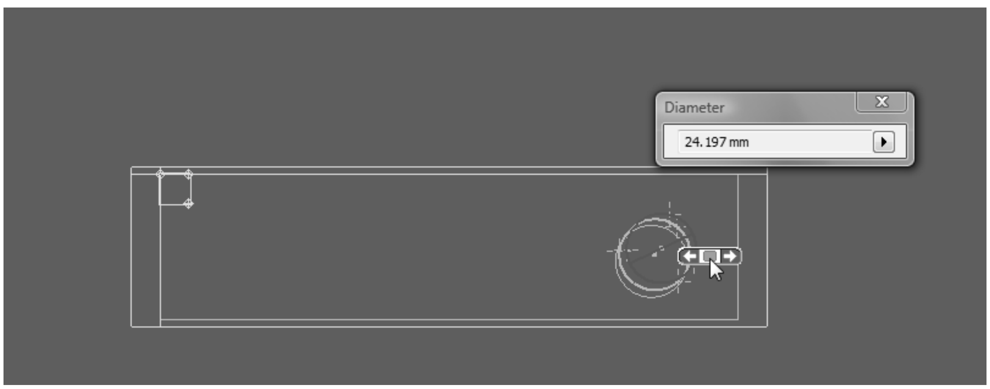


Figure E.58: Measuring one inspection circle.

3. Next select the second measurement circle and note its diameter as in Figure E.59. (To select the circle you may have to *Right Click* ⇒ *Select Other* and then cycle through until the circle is highlighted. Then click the green middle button.) In this case the diameter is 24.742 mm, which also falls within the dimensional size tolerance.

Figure E.59: Measuring the second inspection circle.

Measure the Axis Position

The last step in the inspection process, is to inspect the position of the axis. The starting point of the axis (on the base circle) and the ending point of the axis (on the top circle) must fall within the true position boundary circle. In this case, the circle is located 170 mm from the ***-C-*** datum (or XZ-plane) and 30 mm from the ***-B-*** datum (or YZ-plane). The boundary circle at this location is 5 mm.

1. On the *Model* tab select the *Create 2D Sketch* tool.

2. Select the XY Plane in the *Model Tree.*

3. The screen should still be oriented in the top view, but if not do this now.

4. Select the *Circle* tool and create an arbitrary circle on the XY Plane as in Figure E.60.

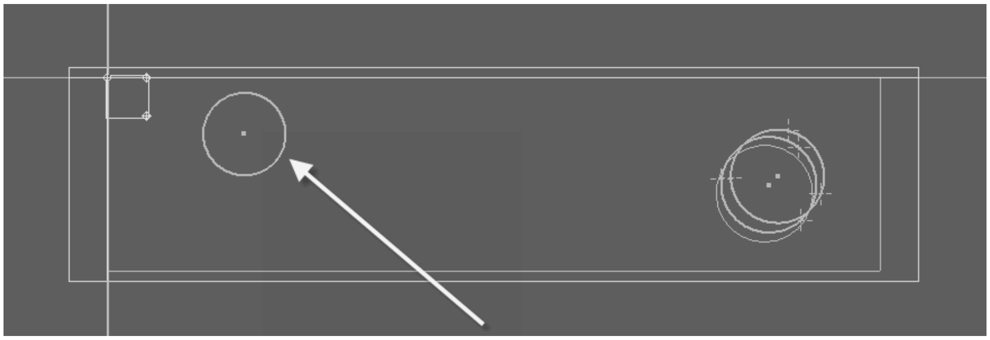


Figure E.60: Sketching an unconstrained circle.

5. Right mouse click and select Done.

6. Select the *Dimension* tool and select the circle that was just created.

7. Enter a value of 5 mm (the tolerance value) for the diameter, as in Figure E.61.

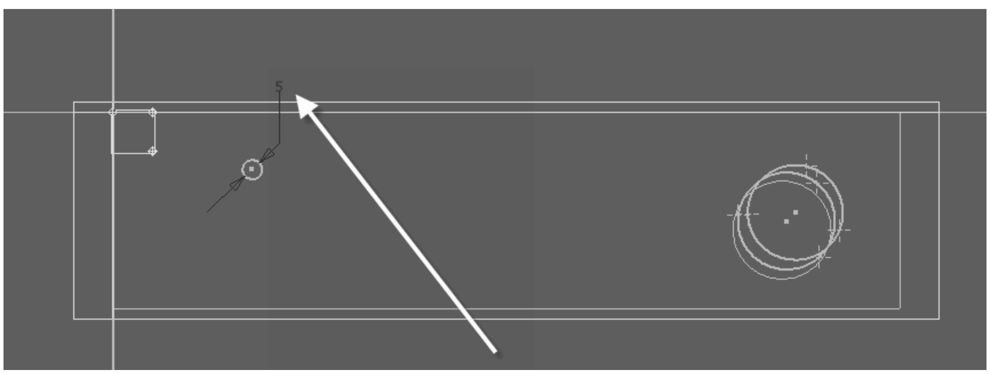


Figure E.61: Defining the diametral tolerance boundary.

8. With the *Dimension* tool still selected, dimension the tolerance boundary circle 170 mm from the XY-Plane and 30 mm from the YZ-Plane, as in Figure E.62.

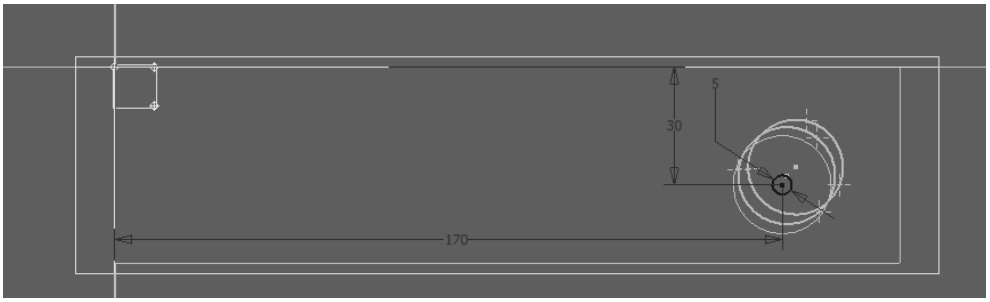


Figure E.62: Defining the tolerance boundary location.

9. The last step is to zoom in and investigate whether the center points of the inspection circles fall within the tolerance boundary circle. If they fall within the circle, the feature meets inspection. In the case of Figure E.63, the axis end points do not fall within the boundary circle. Therefore the feature and the part do not meet inspection.

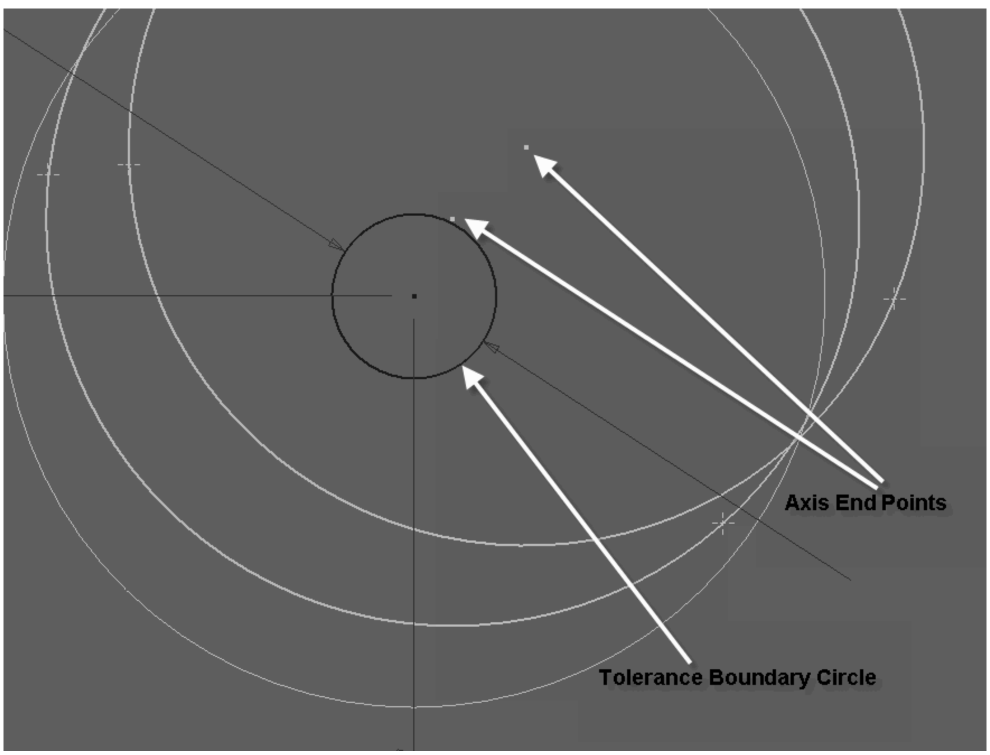


Figure E.63: Examining the inspection points and the tolerance boundary.

APPENDIX F

LAB UNIT ONE-ASSIGNMENT A: INTRODUCTION TO DATUMS AND GD&T INSTRUCTION MANUAL

F.1 One Dimensional Part Variation

Unfortunately, no part in the real world can ever be produced perfectly, because all manufacturing processes inherently contain some variation. Therefore, a designer needs to include *tolerances*, which indicate to what degree imperfections are acceptable. However, even though an engineering drawing can be fully dimensioned and toleranced, there can still be the opportunity for the misinterpretation of a designer's intent when only dimensional tolerances are applied.

F.1.1 Engineering Drawing of Block 1

To demonstrate this fact consider Figure F.1, which is an engineering drawing of a rectangular block.

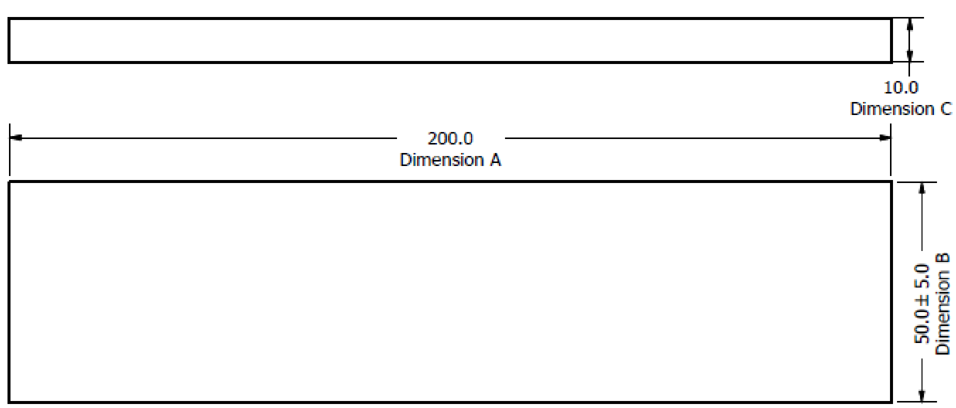


Figure F.1: Engineering drawing of Block 1.

In this drawing, dimension A calls out the length, dimension B calls out the width, and dimension C calls out the thickness. The only tolerance applied in this drawing is the symmetric, bilateral tolerance (i.e. the plus-minus tolerance) on dimension B, while all other dimensions are left basic or without tolerances. Block 1 has been produced according to the drawing in Figure F.1 and can be seen in Figure F.2. The manufacturing error has been greatly exaggerated to allow for better visualization of the variation.

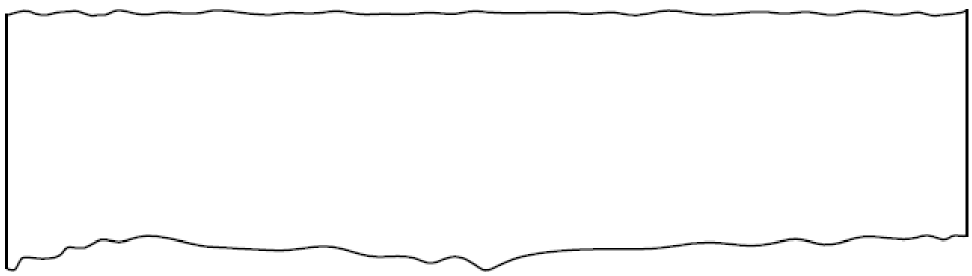


Figure F.2: Front view of Block 1.

F.1.2 Free Inspection of Block 1

Take the electronic digital calipers, like the ones pictured in Figure F.3, and verify they are properly calibrated to zero and the unit of measure is set to millimeters. Then take Block 1, pictured in Figure F.2, and take six linear measurements across its height (dimension B in Figure F.1) on the indicated measurement dots. Record these results in Table G.1. **Take care only to record the measures to the accuracy called out in the drawing and be sure to label the measure with the proper units.**

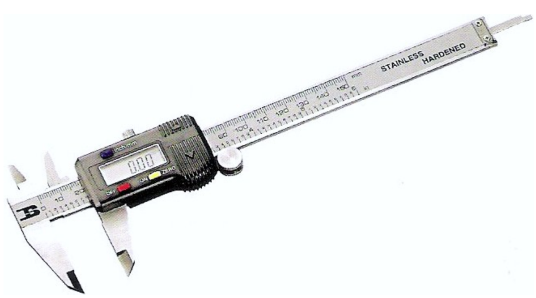


Figure F.3: Electronic calipers.

Note: All the dimensions recorded in Table G.1 should fall between the tolerance limits of 45 mm and 55 mm, since the block was manufactured to meet the drawing requirements of Figure F.1.

F.1.3 Fixed Inspection of Block 1

One of the design requirements Block 1 needs to meet is that the height is no taller than 55.0 mm and no shorter than 45.0 mm when resting on a face with variation, as shown in Figure F.4.

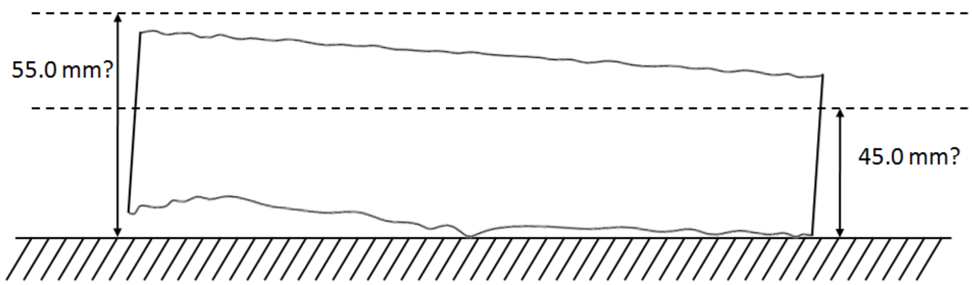


Figure F.4: Free standing Block 1.

To ensure that Block 1 meets this condition, an additional measurement will be needed. During the previous inspection of Block 1, the height of the block was measured relative to itself. During this inspection, the height of each block will be measured from a baseline which will simulate the plane Block 1 will rest on. This simulated surface will be the top measurement rail of the CMM table. In order for all groups to obtain consistent measures, Block 1 needs to be constrained to the top measurement rail identically every time. However, before these measures can be taken, the CMM needs to be interfaced with the computer, so the following features in Autodesk Inventor and the inspection table correspond, as in Table F.1. Follow the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* document found in Appendix B. Once these steps are complete the CMM will be interfaced with Inventor and the two coordinate systems will be aligned. IF THE MICROSCRIBE BASE IS MOVED OR A NEW PART IS OPENED IN INVENTOR, THE CMM WILL NEED TO BE RECALIBRATED.

Table F.1: Coordinate Alignment

Autodesk Inventor		Measurement Table
YZ Plane	⇔	Top Rail
XZ Plane	⇔	Side Rail
XY Plane	⇔	Granite Surface

F.1.4 Constraining Block 1

The next step in the CMM measurement process is to constrain the block to the measurement table. The first step in the constraint process is to lay either of the large surfaces of Block 1 on the table, with the measurement dots facing away from the top rail of the CMM table. Using this large surface as the base, will leave the block stable during measurement. Now Block 1 has only 3 degrees of freedom (DOF) while resting on the table - translation in the X and Y direction rotation about the Z axis. These can be seen in Figure F.5.

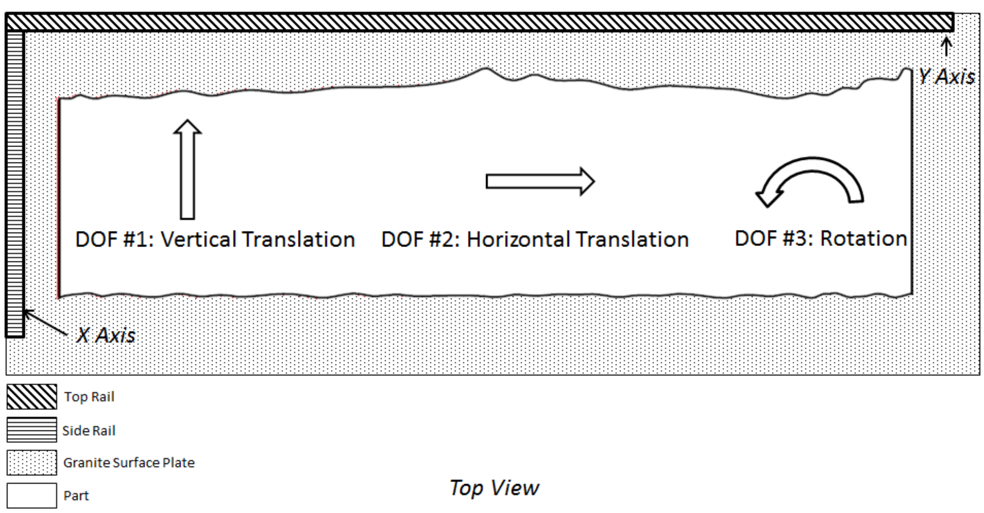
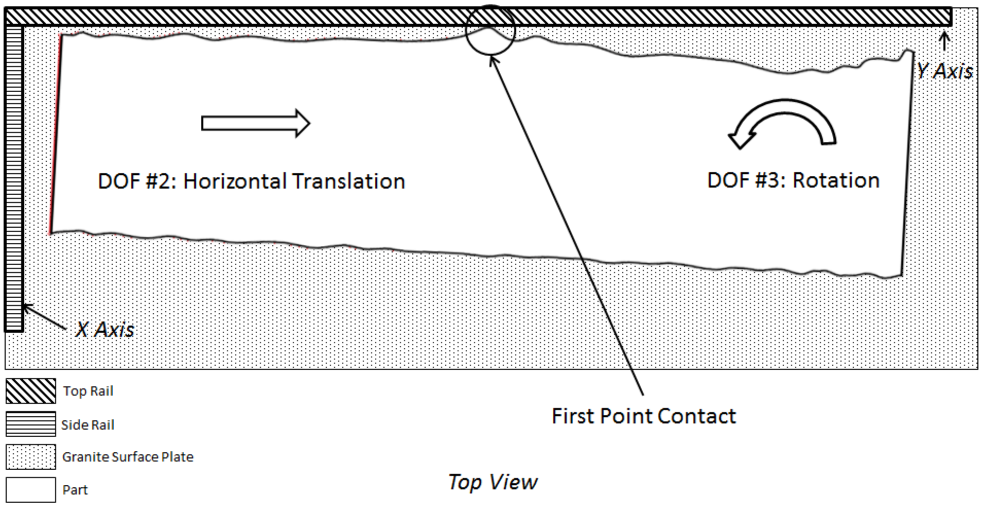
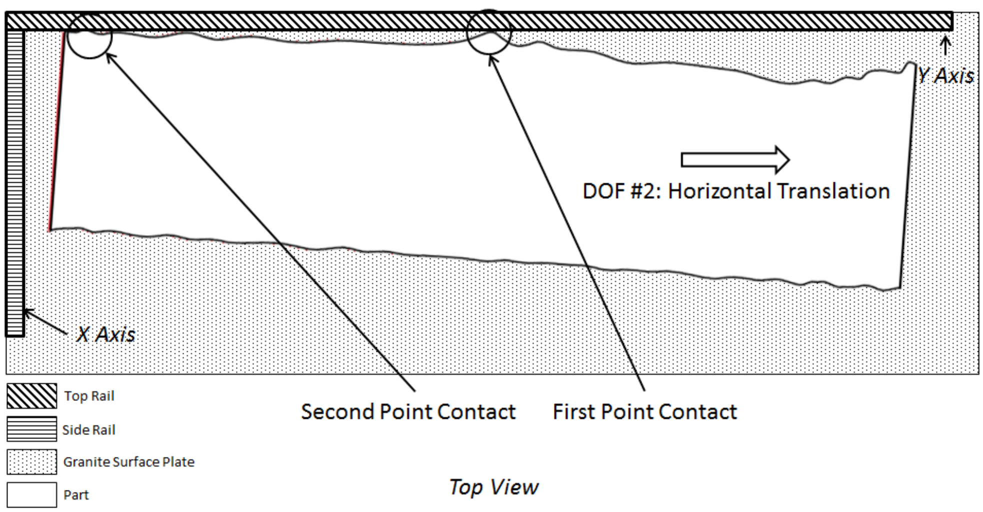


Figure F.5: Degrees of Freedom for resting Block 1.

Next, take the side of the block without the measurement dots and touch the center point of this side to the top measurement rail (see Figure F.6(a)), and then touch a second point on the left of the block to the same measurement rail, see Figure F.6(b).



(a) First point contact constrains vertical translation.



(b) Second point contact constrains rotation.

Figure F.6: Two vertical point contacts constrain translation and rotation.

The first of the point contacts will take away the vertical translation while the second point will constrain the rotation. These two points will form a line which lies in the plane of the top measurement rail and can be seen in Figure F.6.

The final DOF left unconstrained is the horizontal translation along the Y axis. Slide the block to touch the side measurement rail while maintaining two point contact on the top rail, as in Figure F.7. The block is now fully constrained on the measurement table.

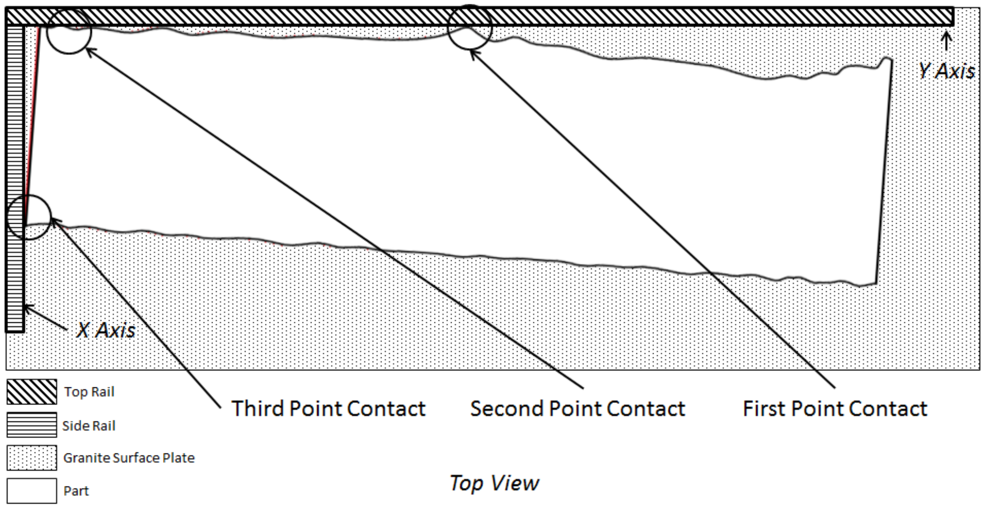


Figure F.7: Three point contacts fully constrain Block 1 to the table.

F.1.5 CMM Measurement

Now that the block is constrained and the CMM is interfaced it is time to take six height measurements on the measurement dots. BE SURE TO KEEP THE BLOCK STEADY AND IN THE SAME CONFIGURATION DURING THE ENTIRE INSPECTION. Record these measures in Table G.2. **Again, take care only to record the measures to the accuracy called out in the drawing and be sure to label the measure with the proper units.** In order to take these points, follow the *Instruction Set: Taking Surface Inspection Points* found in Appendix C. Start a new Inventor file, unhide the XY, YZ, and XZ planes, and use the YZ work plane as the measurement datum.

Note: If constrained properly, not all measurements will fall within the 45.0 mm and 55.0 mm tolerance zone specified in the drawing!

F.1.6 So why should I care?

Even though the engineer knows the block shouldn't be taller than 55.0 mm and the drawing specifies the block should not be taller than 55.0 mm, the block can still be manufactured correctly and still be taller than 55.0 mm! This is because the engineer has not accurately stated the design intent for the part!

- Plus\Minus tolerances only have the ability to specify the FORM (i.e. size) of a part.
- Need to develop a method of communication to specify the FUNCTION (e.g. the height of the part when installed on the ground) on the drawing.
- YOUR ENGINEERING TOOLBOX IS INCOMPLETE!

F.1.7 Datum Order

The engineering drawing of Block 1 has been updated with a datum scheme and can be seen in Figure F.8.

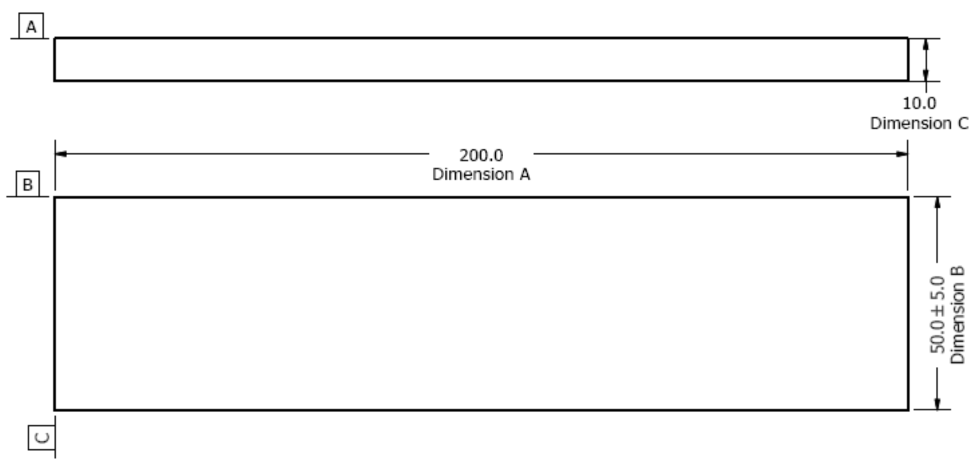


Figure F.8: Engineering drawing of Block 1 with datums.

The order in which the primary, secondary, and tertiary datums are applied in a datum reference frame (DRF) can affect the inspection results of a part. To demonstrate this, Block 1 will be measured again using the CMM, but in a different configuration.

Configuration 2

1. Open Block_1_CMM_Measurement_2.ipt in Autodesk Inventor and save a copy of the part as groupname_1_CMM2.ipt.

2. Repeat the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* (in Appendix B)

3. Constrain the block to the measurement table according to the datum order in Table G.3.

4. During the fixturing of the block record which degrees of freedom are removed by which datum in the third line in Table G.3.

5. Repeat the CMM measurement process on the indicated measurement dots.

6. Use the *Distance* tool under the *Inspect* tab to measure each point and record in Table G.3.

Configuration 3

1. Open Block_1_CMM_Measurement_3.ipt in Autodesk Inventor and save a copy of the part as groupname_1_CMM3.ipt.

2. Repeat the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* (in Appendix B)

3. Constrain the block to the measurement table according to the datum order in Table G.4.

4. During the fixturing of the block, record which degrees of freedom are removed with which datums in the third line in Table G.4.

5. Repeat the CMM measurement process on the indicated measurement dots.

6. Use the *Distance* tool under the *Inspect* tab to measure each point and record in Table G.4.

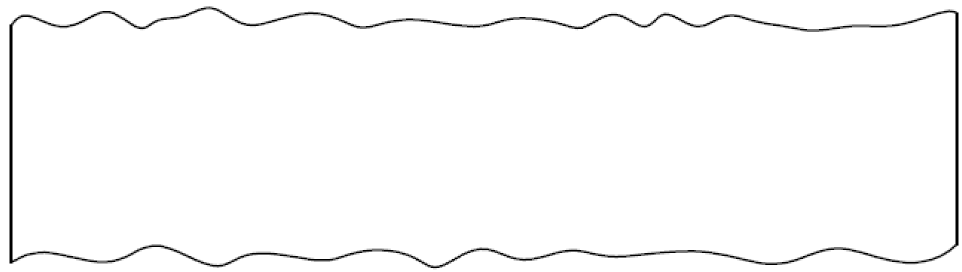
F.1.8 So why should I care?

Because the height measurement of the block changes with the configuration of the block! The order of the primary, secondary, and tertiary datums may not seem to matter much, but in fact they are very important to the inspection results of a part. If a drawing specifies a certain order for a primary, secondary, and tertiary datum, it is usually done to reflect the function of a part! Small details can mean the difference between the success or failure of a part.

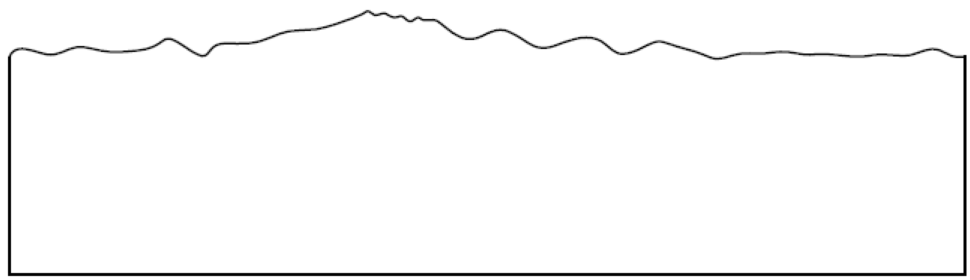
F.2 Tolerance Zones and Variation Location

Now that a method has been established which creates a coordinate system and a constraint process for a part, the next step is to utilize this tool to accurately control a part. However, before this can be done, the concept of tolerance zones and variation location needs to be examined.

Block 2 and Block 3 have been produced which also meet the dimensional tolerance requirements of Figure F.1. They can be seen in Figure F.9.



(a) Front view of Block 2.



(b) Front view of Block 3.

Figure F.9: Front views of blocks manufactured according to the drawing in Figure F.1.

F.2.1 Caliper Inspection of Block 2 and Block 3

Repeat the linear caliper measurements for Block 2 and Block 3 on the indicated measurement dots with the caliper and record the results in Table G.5.

Note: If done correctly, all measures should fall within the 45.0 mm and 55.0 mm tolerance range specified on the engineering drawing.

F.2.2 CMM Inspection of Block 2 and Block 3

Repeat the CMM measurements for Block 2 and Block 3 on the indicated measurement dots. Use ***-A-*** as the primary datum, ***-B-*** as the secondary datum, and ***-C-*** as the tertiary datum.

Block 2 Measurement

1. Open Block_2_CMM_Measurement_1.ipt in Autodesk Inventor and save a copy of the part as groupname_2_CMM1.ipt.

2. Repeat the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* (in Appendix B)

3. Constrain the block to the measurement table and use ***-A-*** as the primary datum, ***-B-*** as the secondary datum, and ***-C-*** as the tertiary datum.

4. Use the *Distance* tool under the *Inspect* tab to complete the measurements.

5. Record measure results into Table G.6.

Block 3 Measurement

1. Open Block_3_CMM_Measurement_1.ipt in Autodesk Inventor and save a copy of the part as groupname_3_CMM1.ipt.

2. Repeat the *Instruction Set: Portable CMM Synchronization with Autodesk Inventor* (in Appendix B)

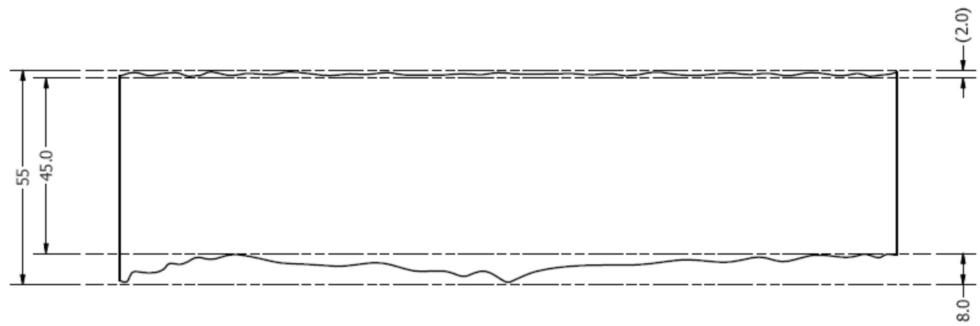
3. Constrain the block to the measurement table and use ***-A-*** as the primary datum, ***-B-*** as the secondary datum, and ***-C-*** as the tertiary datum.

4. Use the *Distance* tool under the *Inspect* tab to complete the measurements.

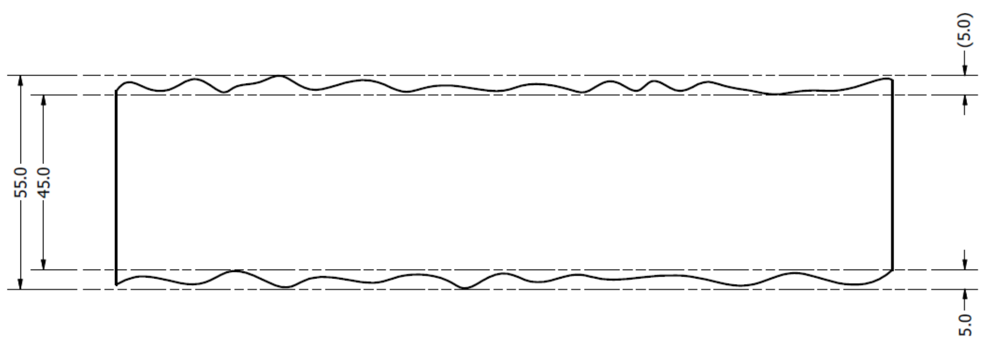
5. Record measure results into Table G.6.

F.2.3 Variation Location

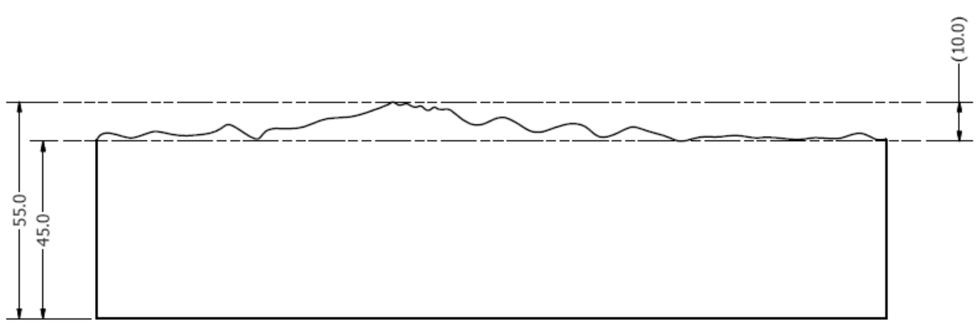
One of the properties of a dimensional tolerance is that the feature variation can occur anywhere within the acceptable limit or *tolerance zone* of the dimension. Examine Figure F.10, which has the maximum and minimum variation dimensioned on each block.



(a) Tolerance zones for Block 1.



(b) Tolerance zones for Block 2.



(c) Tolerance zones for Block 3.

Figure F.10: Variation location for Blocks 1, 2, and 3.

In the instance of Block 1, the variation occurs asymmetrically on the sides of the block, while in Block 2 the variation occurs symmetrically between the two sides. In Block 3 the variation occurs only on one side of the block.

The location of this variation determines the configuration and angular misalignment of each block during CMM inspection. The larger the zone for variation is on one side of the block, the more potential there is for angular misalignment. However, if the variation is evenly split, the angular misalignment during inspection is not as great. In order to control the angular misalignment, which affects how tall the part is while resting freely, the variation location and limits on this variation need to be defined on the engineering drawing. The way this is done will be investigated in the next lab.

APPENDIX G

LAB UNIT ONE-ASSIGNMENT A: INTRODUCTION TO DATUMS AND GD&T WORKSHEET

Table G.1: Caliper Measures for Block 1

Measure	1	2	3	4	5	6
Block 1						

Table G.2: CMM Measurements for Block 1

Measure	Point 1	Point 2	Point 3	Point 4	Point 5	Point 6
Block 1						

Table G.3: Measurements for Block 1-Configuration 2

Case 1	Primary/Granite Surface			Secondary/Top Rail			Tertiary/Side Rail		
Datum	-***B***-			-***A***-			-***C***-		
DOF						-		-	-
Measures							-	-	-

Table G.4: Measurements for Block 1-Configuration 3

Case 2	Primary/Granite Surface			Secondary/Top Rail			Tertiary/Side Rail		
Datum	-***A***-			C			-***B***-		
DOF						-		-	-
Measures							-	-	-

Table G.5: Caliper Measures for Block 2 and Block 3

Measure	1	2	3	4	5	6
Block 1						
Block 2						

Table G.6: CMM Measurements for Block 2 and Block 3

Measure	Point 1	Point 2	Point 3	Point 4	Point 5	Point 6
Block 1						
Block 2						

APPENDIX H

LAB UNIT ONE-ASSIGNMENT A: INTRODUCTION TO DATUMS AND GD&T WORKSHEET KEY

Measurement answers will vary

Table H.1: Caliper Measures for Block 1

Measure	1	2	3	4	5	6
Block 1						

Measurement answers will vary

Table H.2: CMM Measurements for Block 1

Measure	Point 1	Point 2	Point 3	Point 4	Point 5	Point 6
Block 1						

Measurement answers will vary

Table H.3: Measurements for Block 1-Configuration 2

Case 1	Primary/Granite Surface			Secondary/Top Rail			Tertiary/Side Rail		
Datum	***-B-***			***-A-***			***-C-***		
DOF	***Tz***	***Rx***	***Ry***	***Tx***	***Rz***	-	***Ty***	-	-
Measures							-	-	-

Measurement answers will vary

Table H.4: Measurements for Block 1-Configuration 3

Case 2	Primary/Granite Surface			Secondary/Top Rail			Tertiary/Side Rail		
Datum	***-A-***			C			***-B-***		
DOF	***Tz***	***Rx***	***Ry***	***Tx***	***Rz***	-	***Ty***	-	-
Measures							-	-	-

Measurement answers will vary

Table H.5: Caliper Measures for Block 2 and Block 3

Measure	1	2	3	4	5	6
Block 1						
Block 2						

Measurement answers will vary

Table H.6: CMM Measurements for Block 2 and Block 3

Measure	Point 1	Point 2	Point 3	Point 4	Point 5	Point 6
Block 1						
Block 2						

APPENDIX I

LAB UNIT ONE-ASSIGNMENT B: PART INSPECTION INSTRUCTION MANUAL

I.1 Geometric Tolerances

During this next set of exercises, geometric tolerances and their boundaries will be practiced. Two parts and their engineering drawings have been created which contain geometric tolerances, and these can be seen in Figures I.1 and I.2. Use the Microscribe, Autodesk Inventor, and Microsoft Excel Inspection sheets to inspect the produced parts according to the methods presented in the *Geometric Tolerancing Lab Background Information*, found in Appendix E. Use the Autodesk Inventor parts lab_1.1_inspection.ipt and lab_1.2_inspection.ipt to perform the inspection and be sure to create the boundary zones and offset planes where appropriate. Complete the accompanying worksheets in Appendix J and Appendix L to record the data taken.

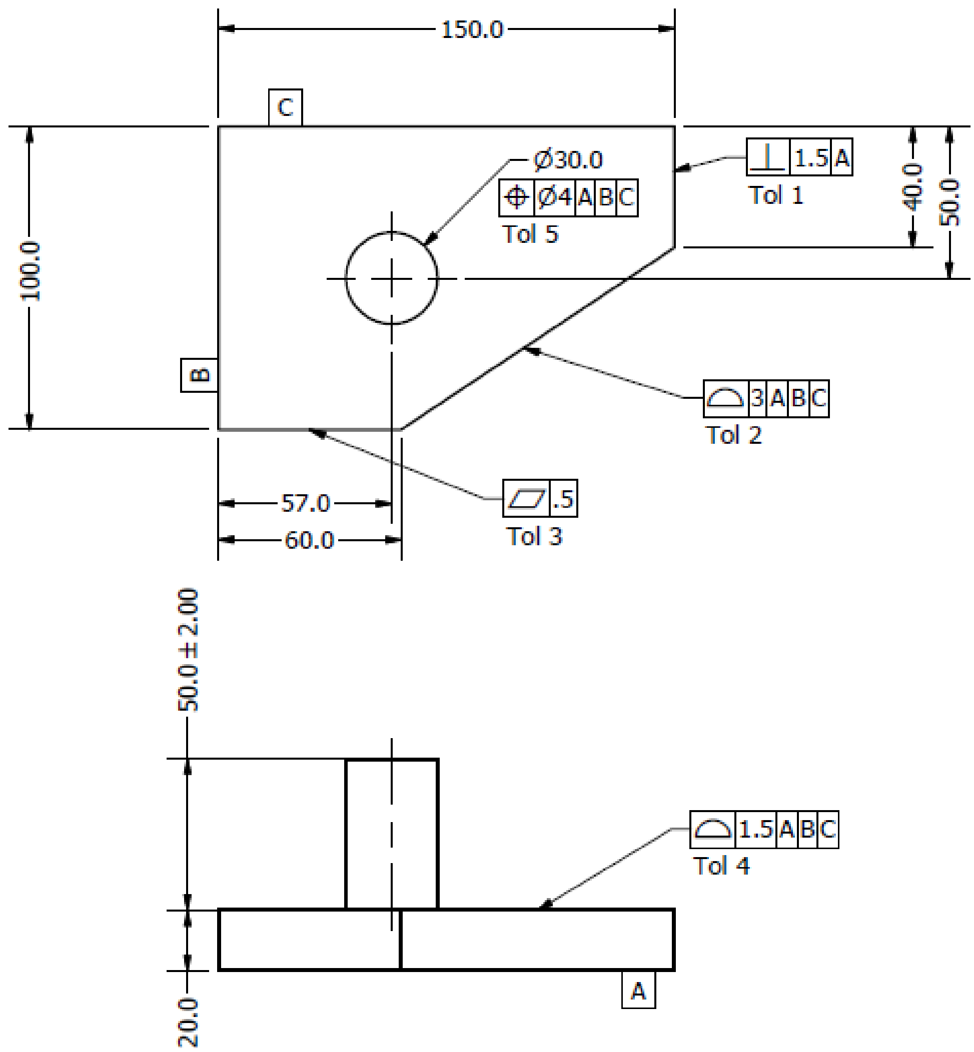


Figure I.1: Engineering drawing of Part 2.

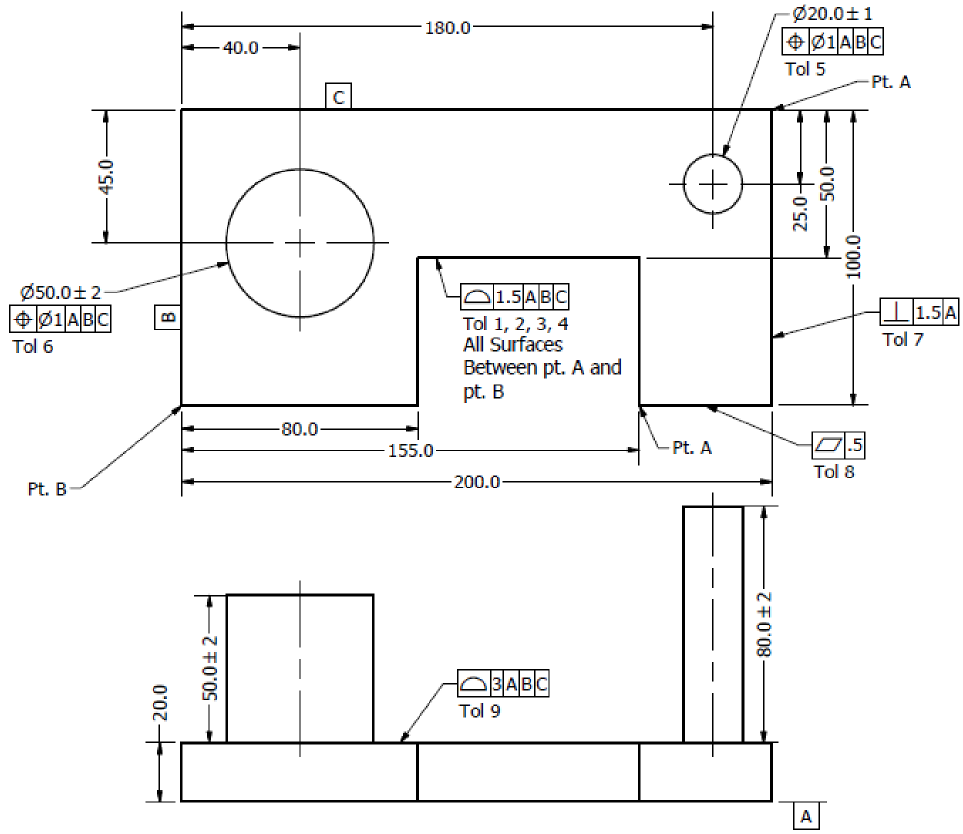


Figure I.2: Engineering drawing of Part 3.

APPENDIX J

LAB UNIT ONE-ASSIGNMENT B: PART 2 INSPECTION WORKSHEET

1. Fill out the following table regarding the tolerances on Part 2.

Tolerance Number	Tolerance Class	Tolerance Type
Tol 1		
Tol 2		
Tol 3		
Tol 4		
Tol 5		

J.0.1 Tol 1 Inspection

2. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

3. What is the perfect feature?

   (a) Axis

   (b) Plane

   (c) Surface

4. What shape does the boundary zone take?

   (a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area

5. Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

6. What is the maximum variation?

J.0.2 Tol 2 Inspection

7. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

8. What is the perfect feature?

(a) Axis

(b) Plane

(c) Surface

9. What shape does the boundary zone take?

(a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area

10. Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

11. What is the maximum variation?

J.0.3 Tol 3 Inspection

12. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

13. What is the perfect feature?

    (a) Axis

    (b) Plane

    (c) Surface

14. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area

15. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

16. What is the maximum variation?

J.0.4 Tol 4 Inspection

17. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

18. What is the perfect feature?
    (a) Axis
    (b) Plane
    (c) Surface
19. What shape does the boundary zone take?
    (a) Cylinder
    (b) Rectangular Prism
    (c) Infinite Planar Area
20. Does the part meet the tolerance requirement?
    (a) Passes
    (b) Fails
21. What is the maximum variation?

J.0.5 Tol 5 Inspection

Primary Datum	Secondary Datum	Tertiary Datum

What is the perfect feature?

(a) Axis

(b) Plane

(c) Surface

What shape does the boundary zone take?

(a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area

Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

APPENDIX K

LAB UNIT ONE-ASSIGNMENT B: PART 2 INSPECTION WORKSHEET KEY

1. Fill out the following table regarding the tolerances on Part 2.

Tolerance Number	Tolerance Class	Tolerance Type
Tol 1	***Orientation***	***Perpendicularity***
Tol 2	***Location***	***Profile***
Tol 3	***Form***	***Flatness***
Tol 4	***Location***	***Profile***
Tol 5	***Position***	***True Position***

K.0.6 Tol 1 Inspection

2. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum
A		

3. What is the perfect feature?

   (a) Axis

   (b) Plane **--*Answer***

   (c) Surface

4. What shape does the boundary zone take?

   (a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area ***Answer***

5. Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

6. What is the maximum variation?

K.0.7 Tol 2 Inspection

7. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum
A	***B***	***C***

8. What is the perfect feature?

(a) Axis

(b) Plane

(c) Surface ***-Answer***

9. What shape does the boundary zone take?

(a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area ***Answer***

10. Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

11. What is the maximum variation?

K.0.8 Tol 3 Inspection

12. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum
NA		

13. What is the perfect feature?

    (a) Axis

    (b) Plane ***-Answer***

    (c) Surface

14. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area ***-Answer***

15. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

16. What is the maximum variation?

K.0.9 Tol 4 Inspection

17. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum
A	***B***	***C***

18. What is the perfect feature?
    (a) Axis
    (b) Plane
    (c) Surface ***-Answer***
19. What shape does the boundary zone take?
    (a) Cylinder
    (b) Rectangular Prism
    (c) Infinite Planar Area ***-Answer***
20. Does the part meet the tolerance requirement?
    (a) Passes
    (b) Fails
21. What is the maximum variation?

K.0.10 Tol 5 Inspection

Primary Datum	Secondary Datum	Tertiary Datum
A	***B***	***C***

What is the perfect feature?

(a) Axis ***-Answer***

(b) Plane

(c) Surface

What shape does the boundary zone take?

(a) Cylinder ***-Answer***

(b) Rectangular Prism

(c) Infinite Planar Area

Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

APPENDIX L

LAB UNIT ONE-ASSIGNMENT B: PART 3 INSPECTION WORKSHEET

1. Fill out the following table regarding the tolerances on Part 3.

Tolerance Number	Tolerance Class	Tolerance Type
Tol 1		
Tol 2		
Tol 3		
Tol 4		
Tol 5		
Tol 6		
Tol 7		
Tol 8		

L.0.11 Tol 1 Inspection

2. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

3. What is the perfect feature?

   (a) Axis

   (b) Plane

   (c) Surface

4. What shape does the boundary zone take?
   (a) Cylinder
   (b) Rectangular Prism
   (c) Infinite Planar Area
5. Does the part meet the tolerance requirement?
   (a) Passes
   (b) Fails
6. What is the maximum variation?

L.0.12 Tol 2 Inspection

7. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

8. What is the perfect feature?
   (a) Axis
   (b) Plane
   (c) Surface
9. What shape does the boundary zone take?
   (a) Cylinder
   (b) Rectangular Prism
   (c) Infinite Planar Area
10. Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

11. What is the maximum variation?

L.0.13 Tol 3 Inspection

12. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

13. What is the perfect feature?

(a) Axis

(b) Plane

(c) Surface

14. What shape does the boundary zone take?

(a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area

15. Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

16. What is the maximum variation?

L.0.14 Tol 4 Inspection

17. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

18. What is the perfect feature?

    (a) Axis

    (b) Plane

    (c) Surface

19. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area

20. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

21. What is the maximum variation?

L.0.15 Tol 5 Inspection

Primary Datum	Secondary Datum	Tertiary Datum

What is the perfect feature?

(a) Axis

(b) Plane

(c) Surface

What shape does the boundary zone take?

(a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area

Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

L.0.16 Tol 6 Inspection

Primary Datum	Secondary Datum	Tertiary Datum

What is the perfect feature?

(a) Axis

(b) Plane

(c) Surface

What shape does the boundary zone take?

(a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area

Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

L.0.17 Tol 7 Inspection

22. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

23. What is the perfect feature?

    (a) Axis

    (b) Plane

    (c) Surface

24. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area

25. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

26. What is the maximum variation?

L.0.18 Tol 8 Inspection

27. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

28. What is the perfect feature?

    (a) Axis

    (b) Plane

    (c) Surface

29. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area

30. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

31. What is the maximum variation?

L.0.19 Tol 9 Inspection

32. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

33. What is the perfect feature?

    (a) Axis

    (b) Plane

    (c) Surface

34. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area

35. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

36. What is the maximum variation?

APPENDIX M

LAB UNIT ONE-ASSIGNMENT B: PART 3 INSPECTION WORKSHEET KEY

1. Fill out the following table regarding the tolerances on Part 3.

Tolerance Number	Tolerance Class	Tolerance Type
Tol 1	***Location***	***Profile***
Tol 2	***Location***	***Profile***
Tol 3	***Location***	***Profile***
Tol 4	***Location***	***Profile***
Tol 5	***Position***	***True Position***
Tol 6	***Position***	***True Position***
Tol 7	***Orientation***	***Perpendicularity***
Tol 8	***Form***	***Flatness***
Tol 9	***Location***	***Profile***

M.0.20 Tol 1 Inspection ***(Answers for tolerances 1 - 4 will be identical)***

2. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum
A	***B***	***C***

3. What is the perfect feature?

   (a) Axis

(b) Plane

(c) Surface ***-Answer***

4. What shape does the boundary zone take?

(a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area ***-Answer***

5. Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

6. What is the maximum variation?

M.0.21 Tol 2 Inspection

7. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

8. What is the perfect feature?

(a) Axis

(b) Plane

(c) Surface

9. What shape does the boundary zone take?

(a) Cylinder

(b) Rectangular Prism

(c) Infinite Planar Area

10. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

11. What is the maximum variation?

M.0.22 Tol 3 Inspection

12. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

13. What is the perfect feature?

    (a) Axis

    (b) Plane

    (c) Surface

14. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area

15. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

16. What is the maximum variation?

M.0.23 Tol 4 Inspection

17. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum

18. What is the perfect feature?

    (a) Axis

    (b) Plane

    (c) Surface

19. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area

20. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

21. What is the maximum variation?

M.0.24 Tol 5 Inspection

Primary Datum	Secondary Datum	Tertiary Datum
A	***B***	***C***

What is the perfect feature?

(a) Axis ***-Answer***

(b) Plane

(c) Surface

What shape does the boundary zone take?

(a) Cylinder ***-Answer***

(b) Rectangular Prism

(c) Infinite Planar Area

Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

M.0.25 Tol 6 Inspection

Primary Datum	Secondary Datum	Tertiary Datum
A	***B***	***C***

What is the perfect feature?

(a) Axis ***-Answer***

(b) Plane

(c) Surface

What shape does the boundary zone take?

(a) Cylinder ***-Answer***

(b) Rectangular Prism

(c) Infinite Planar Area

Does the part meet the tolerance requirement?

(a) Passes

(b) Fails

M.0.26 Tol 7 Inspection

22. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum
A		

23. What is the perfect feature?

    (a) Axis

    (b) Plane ***-Answer***

    (c) Surface

24. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area ***-Answer***

25. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

26. What is the maximum variation?

M.0.27 Tol 8 Inspection

27. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum
NA		

28. What is the perfect feature?

    (a) Axis

    (b) Plane ***-Answer***

    (c) Surface

29. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area ***-Answer***

30. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

31. What is the maximum variation?

M.0.28 Tol 9 Inspection

32. What are the primary, secondary, and tertiary datums? If not applicable put NA.

Primary Datum	Secondary Datum	Tertiary Datum
A	***B***	***C***

33. What is the perfect feature?

    (a) Axis

    (b) Plane

    (c) Surface ***-Answer***

34. What shape does the boundary zone take?

    (a) Cylinder

    (b) Rectangular Prism

    (c) Infinite Planar Area ***-Answer***

35. Does the part meet the tolerance requirement?

    (a) Passes

    (b) Fails

36. What is the maximum variation?

APPENDIX N

LAB UNIT TWO - LECTURE: USING GD&T TO EXPRESS DESIGN INTENT

Now that the definition, history, and working knowledge of geometric dimensioning and tolerancing have been presented and practiced, it is now time to apply the fundamentals of GD&T in a design situation. The goals of this laboratory are to: 1) Identify design criteria given in the problem statement, 2) Categorize which requirements can be controlled through dimensioning and tolerancing, 3) Apply proper GD&T so that the engineering drawing reflects design intent.

N.1 Example 1: Widget Blanks

Your company, the Widget Manufacturing Company, is in the process of finding a supplier to produce blanks for your proprietary widget manufacturing process. It is your responsibility to create an engineering drawing with an appropriate tolerance scheme, so the supplier can create parts as accurately as possible. You have been given the engineering drawing of the blank with only basic dimensions included. This can be seen in Figure N.1

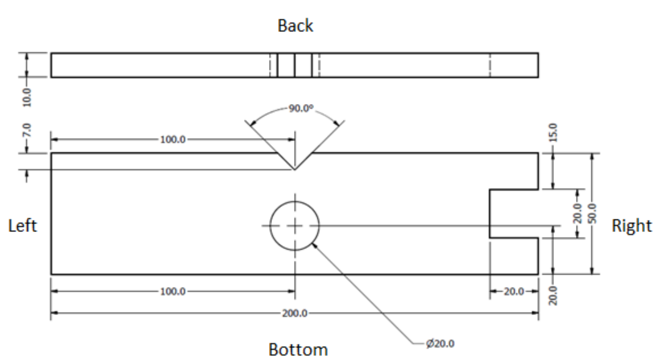


Figure N.1: A drawing of the widget blank with basic dimensions. Labels have been added to the sides of the blank for easy reference.

N.1.1 Investigation

Before attempting to annotate the drawing with the appropriate tolerances, you decide to do some investigation to see how the proprietary widget manufacturing process works.

Schematic

While asking around the office, you have located the following schematic explaining how the widget manufacturing process works:

1. A blank is inserted into the proprietary widget manufacturing machine, see Figure N.2. The triangular notch needs to be oriented on top side of the blank and the rectangular edge needs to be on the right side of the blank.

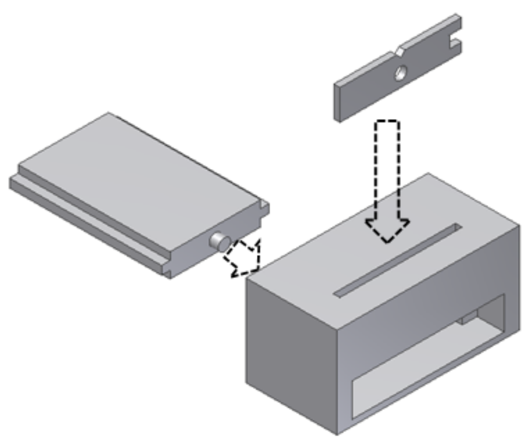


Figure N.2: Step one of the widget manufacturing process.

2. After the blank is inserted, it settles at the bottom of the widget machine, see Figure N.3

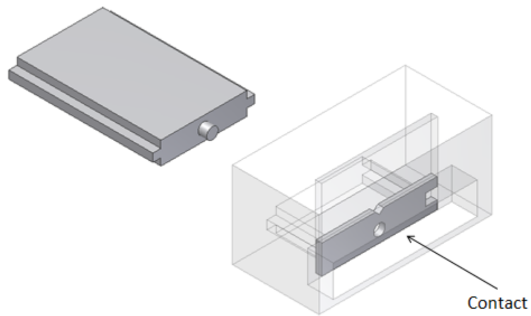


Figure N.3: Step two of the widget manufacturing process.

3. A pusher bar is then driven through the machine until it engages the blank with its guide post, see Figure N.4.

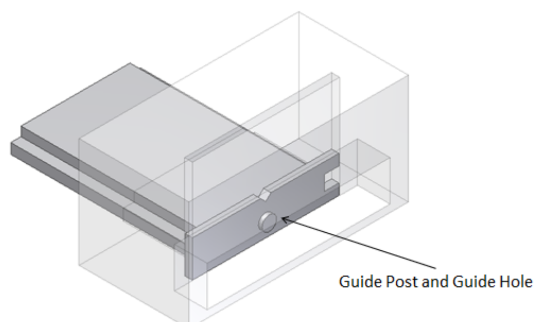


Figure N.4: Step three of the widget manufacturing process.

4. The pusher bar then continues through the machine until the blank passes through the machine and is manufactured into a widget, see Figure N.5.

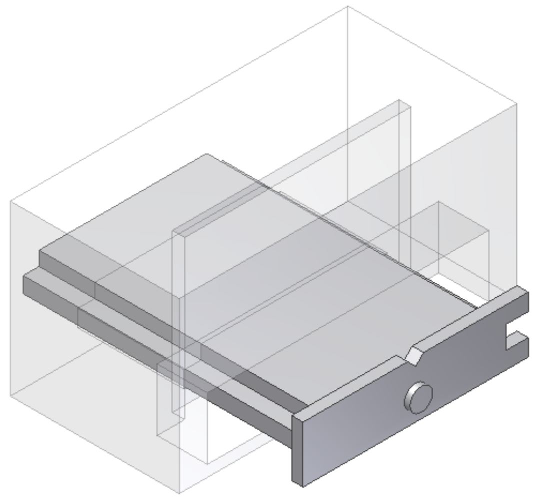


Figure N.5: Step four of the widget manufacturing process.

Documentation

After a visit to the manufacturing floor, you have discovered documentation about the manufacturing process and the requirements for the widget blanks.

The list of requirements for the blank include:

- The interface between the blank bottom and the machine base must not have any gaps larger than 1 mm. The machine base has been manufactured flat within a 0.01 mm surface finish (since this is an order of magnitude greater, it can be assumed perfectly flat). If this interface is not met, then the rest of the process cannot proceed.
- The left side surface must remain perpendicular to the bottom face within 1°.
- The back surface must remain perpendicular within 2° to both the bottom face and the left side surface.
- To avoid any possible interference during step 2 of the manufacturing process:
    - The blank must not be longer than 200.5 mm and not narrower than 199.5.
    - The blank must not be thicker than 10.25 mm and not thinner than 9.75 mm.
    - The blank must not be taller than 55 mm and not shorter than 45 mm.

- To ensure good engagement between the pusher bar and the blank:
  - The guide hole needs be positioned from the left surface and the bottom surface somewhere within a radius of 1.5 mm located around the perfect location.
  - The guide hole needs to have a minimum size of 20 mm and a maximum size of 25 mm.
  - The axis of the guide hole needs to have a maximum angular misalignment 20° with respect to the back surface.
- The two notches are only used as indicators to ensure proper orientation when the blank is inserted into the machine and are not functional features. The only critical factor on these notches is their general shape and location.
  - The centerline of the triangular notch needs to be located within 5 mm of the horizontal center of the part. The angular shape of the notch may vary up to 4°. The depth of the notch may vary 2 mm in either direction.
  - The rectangular notch needs to be located between 10 mm and 20 mm of the top of the part. The depth and height of the notch may vary 2 mm in either direction.

N.2 Annotating the Engineering Drawing

Now that all the background information about the widget blank has been considered, it is now time to begin annotating the drawing. It is always important to consider the fit, form, and function of a part before beginning to dimension and tolerance it. This way design intent can be accurately conveyed through the drawing.

N.2.1 Assigning a Datum Scheme

The first step in annotating an engineering drawing is to assign a datum scheme, which will be used as reference baselines for the part. As mentioned previously, datums are used to indicate important features in an engineering design, which is why it is so important to investigate the fit, form, and function of a part before completing the drawing.

Assigning the *-A-* Datums

After reviewing the process documentation, it is evident from the first step that:

The interface between the blank bottom and the machine base must not have any gaps larger than 1 mm. The machine base has been manufactured flat within a 0.01 mm surface finish (since this is an order of magnitude greater, it can be assumed perfectly flat). If this interface is not met, then the rest of the process cannot proceed.

This is an indicator that the contact with bottom surface is the most important function of the blank. If this feature is not correct, then none of the other features matter and the process cannot proceed. Therefore this will be assigned the ***-A-*** datum as seen in Figure N.6.

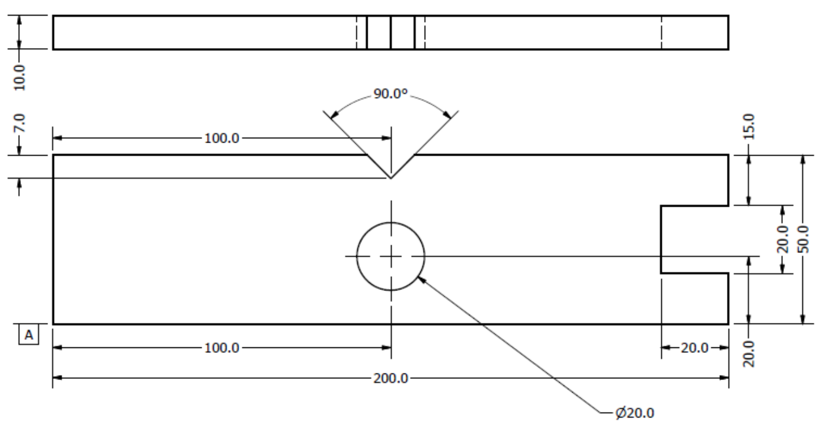


Figure N.6: Assigning the ***-A-*** datum.

By assigning the ***-A-*** datum as the planar contact between the machine base and the blank, three degrees of freedom have been removed from the part. These degrees of freedom are the translation in the vertical direction and rotations about the machine base plane.

Assigning the ***-B-*** and ***-C-*** Datum

The next step in assigning the datum scheme is to scan for information describing the remaining surfaces that can serve as datums. After reviewing the process documentation, the following statements are found:

- The left side surface must remain perpendicular to the bottom face within 1°.
- The back surface must remain perpendicular within 2° to both the bottom face and the left side surface.

Since these two surfaces specifically have requirements placed on them for the success of the process, they make good candidates to serve as the ***-B-*** datum and ***-C-*** datum. Additionally, these two surfaces and the bottom surface are all nominally mutually perpendicular and therefore will make a good baseline system.

After further examination, the left surface only references the bottom surface, while the back surface references both the bottom surface and the smaller side surface. Since the left surface references the primary datum, it is a good choice for the secondary datum. This leaves the back surface a as the only choice for a tertiary datum. According to the documentation, it needs to reference both the primary datum and the secondary datum. In this example, the secondary datum was labeled as ***-B-*** and the tertiary datum was labeled as ***-C-***. While it is not a condition that the primary, secondary, and tertiary datums are in alphabetical order (i.e. ***-A-***, ***-B-***,***-C-***), it typically makes the drawing clearer. The full datum scheme can be seen in Figure N.7.

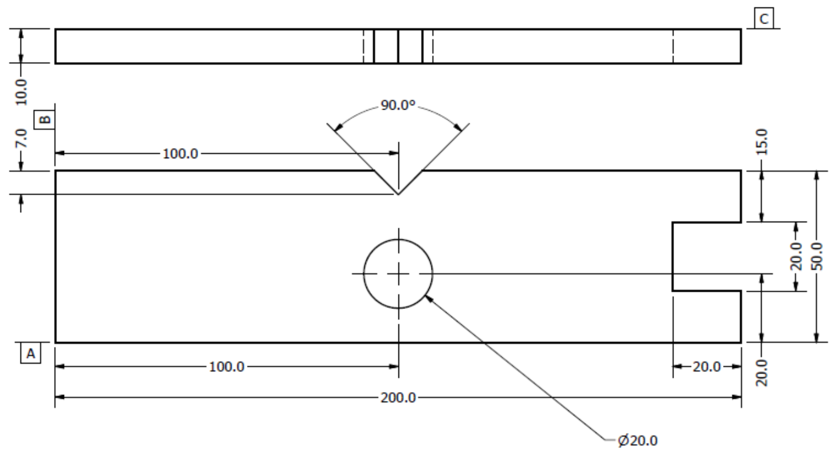


Figure N.7: The full datum scheme.

The ***-B-*** datum (the left surface), removes two additional degrees of freedom: translation along the machine slot length and rotations along the direction of insertion. The ***-C-*** datum removes the last remaining degree of freedom, translations along the slot width. By fully defining a datum scheme, all six degrees of the blank are constrained.

N.2.2 Defining the Datum Scheme Relationship

In a Cartesian coordinate system, it is implied that the XY, YZ, and XZ planes are all mutually perpendicular to one another. This is not true with datum schemes, and the relationships must be defined.

Defining the Primary Datum

The first step is to define the form of the primary datum. According to the process documentation:

The interface between the blank bottom and the machine base must not have any gaps larger than 1 mm. The machine base has been manufactured flat within a 0.01 mm surface finish (since this is an order of magnitude greater, it can be assumed perfectly flat).

This can be pictorially seen in Figure N.8.

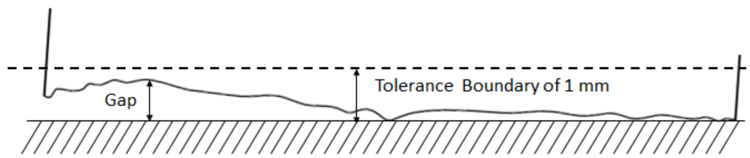


Figure N.8: The maximum gap between the blank and the machine base must be less than 1 mm in length.

Since ***-A-*** is also the primary datum, no other datums may be referenced when defining the shape (or waviness) of ***-A-***. As a result, the only descriptor available is the *Form* category of geometric tolerances. In this specific case, the requirement is that the surface must not have any gaps larger than 1 mm. This can be defined using a *surface flatness* geometric tolerance, with a tolerance boundary of 1 mm. This can be seen in Figure N.9.

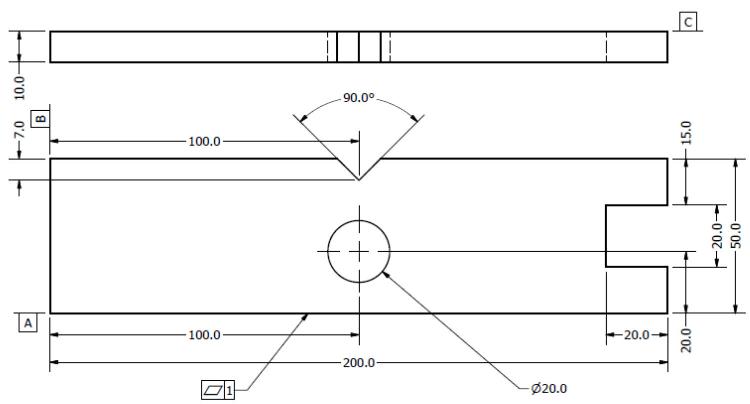


Figure N.9: A flatness tolerance applied to datum ***-A-***.

Additionally, the *Form* tolerance category does not ever require any additional datum reference features.

Defining the Secondary Datum

The next step, is to relate the ***-B-*** datum to the ***-A-*** datum. It is defined in the process details that: *The left side surface must remain perpendicular to the bottom face within 1°*

The only relationship between the ***-A-*** datum and the ***-B-*** datum is an orientation, which is 90°. The requirement is that the ***-B-*** datum varies no more than 1° from the ***-A-*** datum.

This requirement can be pictorially seen in Figure N.10.

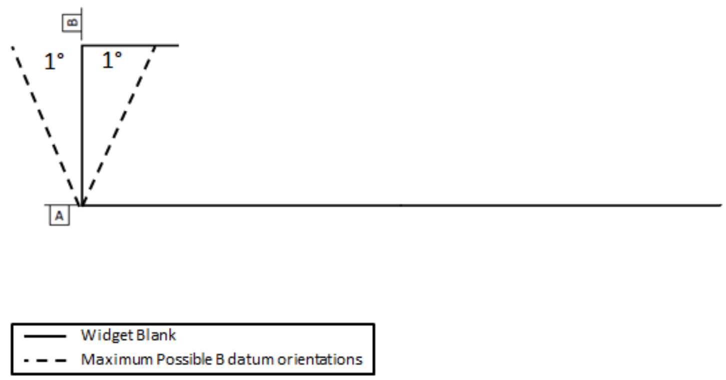


Figure N.10: The maximum misalignment for the ***-B-*** datum.

Due to this relationship, a perpendicularity tolerance needs to be applied to the ***-B-*** datum and the ***-A-*** datum needs to be used as the primary reference for this geometric tolerance. This can be seen in Figure N.11

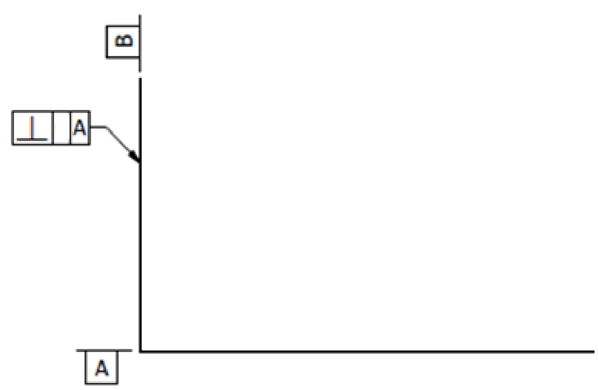


Figure N.11: Perpendicularity tolerance applied to ***-B-***.

The next step is to translate the ±1° angularity requirement to the linear tolerance boundary zone value in millimeters. This is done by taking into consideration the maximum surface misalignment as well as the maximum and minimum height variation, which can be pictorially seen in Figure N.12. According to the process documentation: *The blank must not be taller than 55 mm and not shorter than 45 mm.*

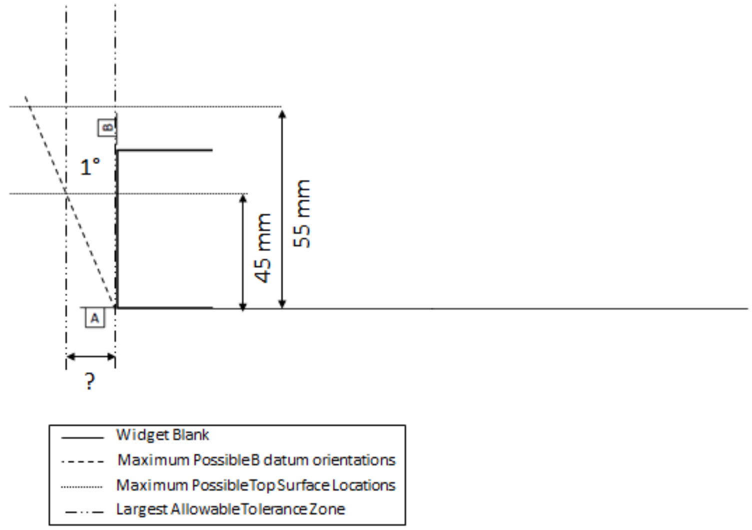


Figure N.12: Translating the angularity requirement to a linear tolerance value.

The limiting case of the tolerance boundary occurs with the shortest blank at the maximum angular misalignment. If the shortest blank has no more than a 1° angular misalignment, then the tallest blank will have no more than a 1° in the same linear tolerance boundary. At this point, calculation of the value becomes a simple trigonometry problem, as seen in Figure N.13. **Note: only one of the maximum misalignment cases must be considered. If both are considered the zone would be twice as large as necessary.**

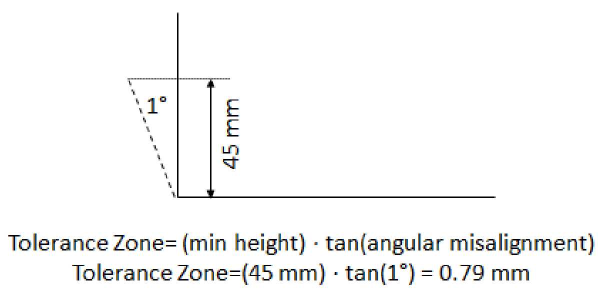


Figure N.13: Calculating the perpendicularity tolerance value.

This calculated value of 0.79 mm is now entered as the perpendicularity tolerance value and the ***-A-*** datum and ***-B-*** datum are now fully defined with respect to each other and 5 degrees of freedom have been removed. This can be seen in Figure N.14.

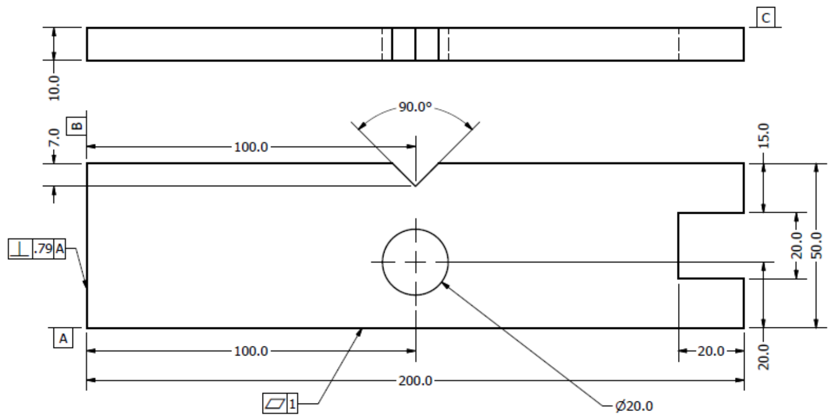


Figure N.14: The fully toleranced ***-B-*** datum.

Defining the Tertiary Datum

In the process details it states:

The back surface must remain perpendicular within 2° to both the bottom face and the left side surface.

Examining the relationship between the ***-A-*** datum and the ***-C-*** datum, it appears that it is analogous to the relationship between the ***-A-*** datum and the ***-B-*** datum. The only difference is that now the allowable angular variation is 2° instead of 1° and the problem is oriented from the side view instead of the front view. This calculation can be seen in Figure N.15.

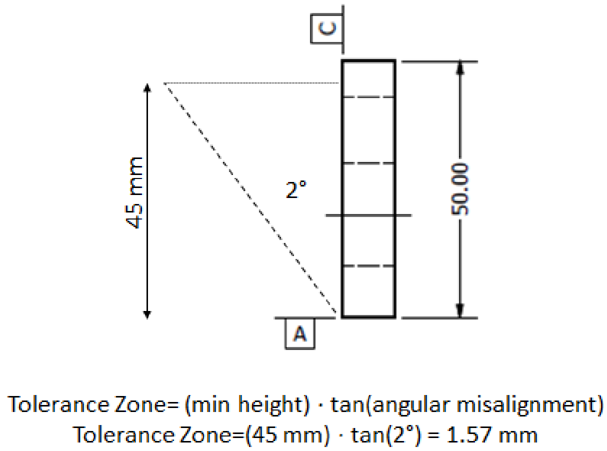


Figure N.15: Calculating the tolerance zone from datum ***-C-*** to datum ***-A-***.

Before this can be entered as the tolerance value, the 2° requirement must be check against datum ***-B-*** as well. This can be pictorially seen in Figure N.16. The instructions have stated: *The blank must not be longer than 200.5 mm and not narrower than 199.5.*

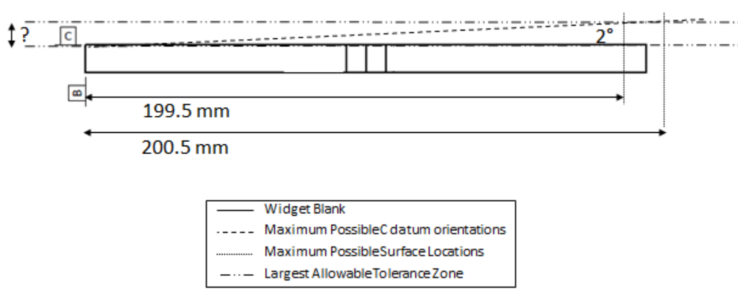


Figure N.16: Pictorially the tolerance zone from datum ***-C-*** to datum ***-B-***.

This again breaks down into a simple trigonometric problem that can be seen in Figure N.17.

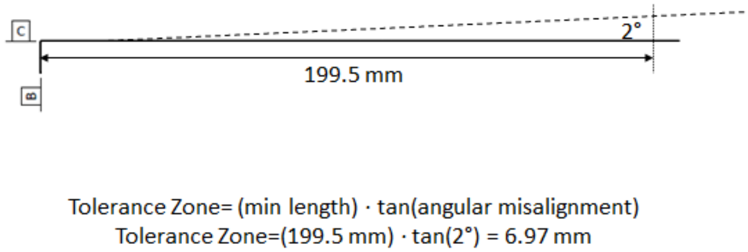


Figure N.17: Calculating the tolerance zone from datum ***-C-*** to datum ***-B-***.

The tolerance zone limit for datum ***-C-*** to datum ***-A-*** is 1.57 mm, while the tolerance zone limit for datum ***-C-*** to datum ***-B-*** is 6.97 mm. Since the value of 1.57 mm will be sufficient for both conditions, it will be entered into the perpendicularity tolerance which is applied to datum ***-C-***. This can be seen in Figure N.18.

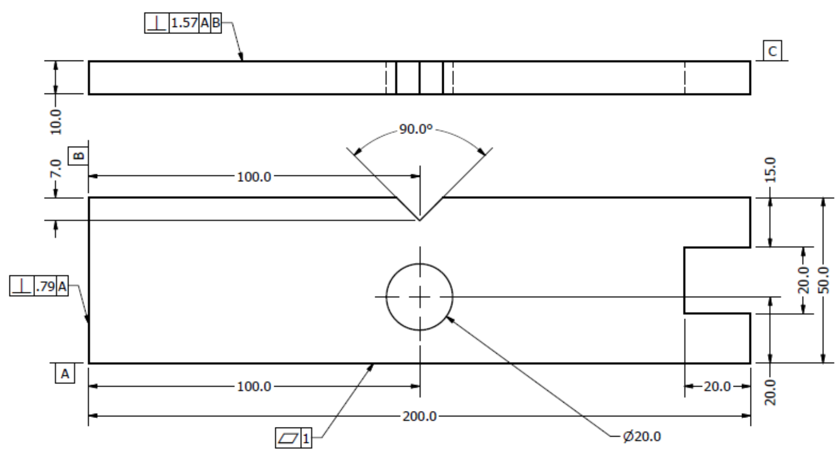


Figure N.18: The datum scheme relationship is now fully defined.

In order to independently relate datum -B- to datum -A- and datum -C-, a tolerance refinement would need to be used. This is a more advanced topic and will not be covered. Additionally, a surface profile tolerance could be used instead of a perpendicularity tolerance and would have achieved the same results.

N.2.3 Controlling the Remaining Functional Surfaces

Now that the baseline system has been established, the height, width, and depth of the part need to be controlled to ensure the blank functions without interferences. The opening on the machine base which the blank fits into is 201 mm wide x 55 mm tall x 11 mm deep. This can be seen in Figure N.19.

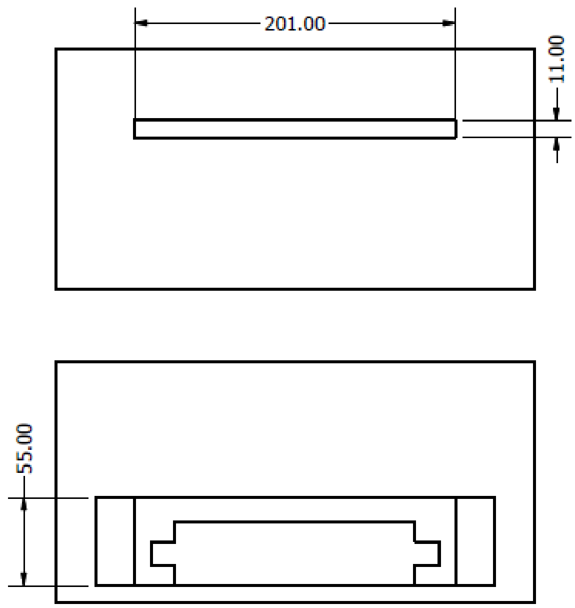


Figure N.19: The critical dimensions of the machine base.

Controlling the Width

In order for the blank to be inserted properly, it must be thinner than the opening of the widget machine. According to the process documentation: *The blank must not be thicker than 10.25 mm and not thinner than 9.75 mm.*

The features that form the width of the blank, are back surface (datum ***-C-***) and the front surface. The requirements placed on the blank create an allowable zone of 0.5 mm. In the worst case, the back surface of the blank would be touching the back of the slot. To avoid interference in this worst case, the front surface must measure narrower than 10.25 mm, when measured against datum ***-C-***. This scenario can be seen in Figure N.20.

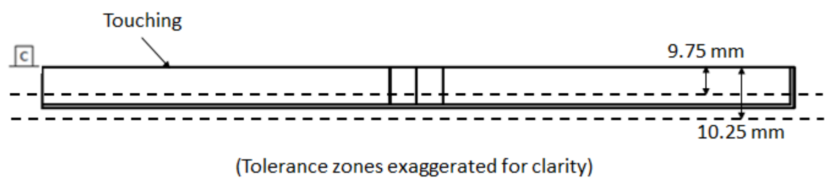


Figure N.20: The schematic for checking the thickness of the blank.

This set of requirements can easily be controlled with a *surface profile* geometric tolerance. In this case, the tolerance value is equal to the boundary zone of 0.5 mm, which is centered about the perfect surface located at 10 mm. Since datum ***-C-*** was the only datum used to set the thickness, it is the only datum needed in the datum reference frame (while additional datums could be added, it is best to avoid adding unnecessary datum references). The updated drawing can be seen in Figure N.21.

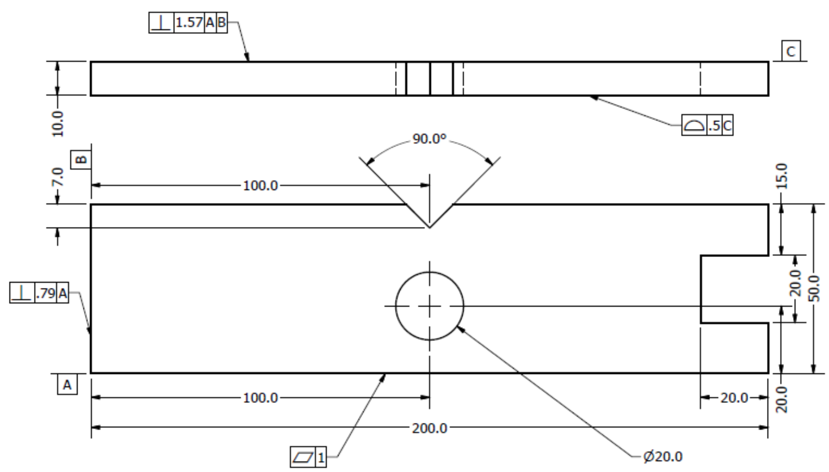


Figure N.21: Profile of a surface applied to the width.

Controlling the Length

During the second step of the operation, interference will occur when if blank is larger than the machine base opening when resting in the worst positional case. In the worst positional case, the blank is touching on three surfaces. The first surface is datum ***-C-*** touching the back of the machine slot and datum ***-A-*** and datum ***-B-*** can be seen touching in Figure N.22:

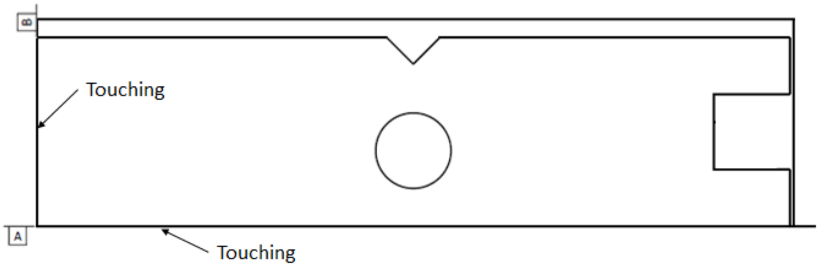


Figure N.22: The worst positional case for the blank.

According to the process documentation: *the blank must not be longer than 200.5 mm and not narrower than 199.5.* This control could be achieved using a ± dimensional tolerance of 0.5 mm, but this does not take into account any variation from the ***-A-***, ***-B-***, or ***-C-*** surface (recall the difference between the caliper vs. CMM measurement from laboratory exercise 1.)

To signify to the manufacturer that the blank must meet the functional requirement using a baseline measurement system, again a *surface profile* geometric tolerance with the datum references should be employed. According to the requirement in the process documentation, the maximum location of the surface should occur at 200.5 mm and the minimum location of the surface should occur at 199.5 mm. This can be translated into a tolerance boundary zone of 1 mm, which is centered on the perfect surface located at 200 mm. An application of this tolerance can be seen in Figure N.23. **Note: the tolerance needs to be applied to both surfaces opposite of datum -*B*-.**

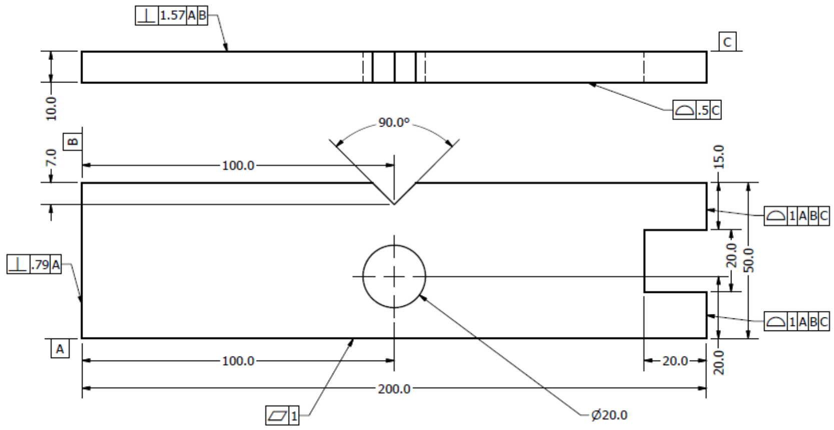


Figure N.23: Profile of a surface applied to the length.

Controlling the Height

According to the process documentation: *The blank must not be taller than 55 mm and not shorter than 45 mm.* This situation, exactly like the other surfaces, will require a *surface profile* geometric tolerance applied to the same datum scheme. This time, the tolerance boundary zone is 10 mm and is centered at the surface 50 mm opposite datum **-*A*-**. The updated drawing can be seen in Figure N.24. **Note: for clarity the tolerance is applied to both surfaces opposite of datum -*A*-. However, when applied to one surface it is assumed to apply to the second surface as well.**

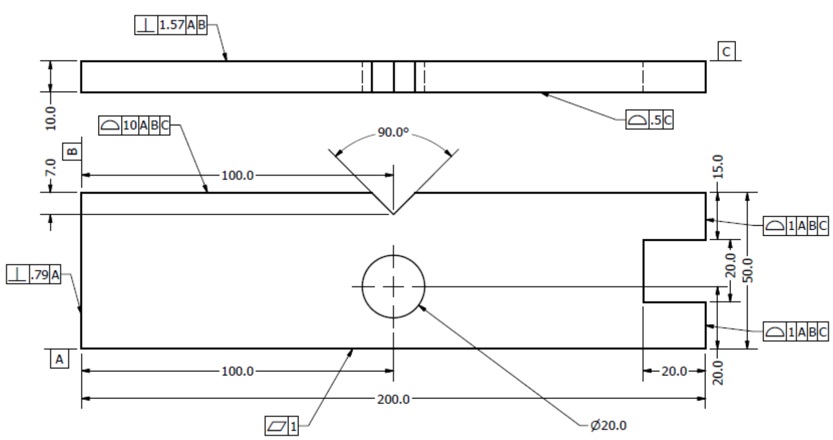


Figure N.24: Profile of a surface applied to the height.

N.2.4 Controlling the Remaining Functional Feature

Now that the general shape or form of the blank has been defined, the only features left to define are the guide hole, triangular notch, and the rectangular notch.

Controlling the Guide Hole

Again after reviewing the process documentation, the following requirements for the guide hole are found:

- *The guide hole needs be positioned from the left surface and the bottom surface somewhere within a radius of 1.5 mm located around the perfect location.*
- *The guide hole needs to have a minimum size of 20 mm and a maximum size of 25 mm.*
- *The axis of the guide hole needs to have a maximum angular misalignment 20° with respect to the large back face.*

The documentation is stating that the position, orientation, and size of the guide hole needs to be controlled. If only position and size needed to be controlled, a standard dimensional tolerance could be used to define this feature. However because orientation is involved along with position, a *true position* geometric tolerance should be used.

The maximum and minimum positions for the center line of the hole can be mapped out for both the vertical and horizontal directions. This boundary can then be translated into a diametral tolerance zone, by finding out the distance between the maximum and the minimum points. In this case, the maximum distance between each set of points is 3 mm, which translates into a tolerance zone of diameter 3 mm. This can be seen in Figure N.25.

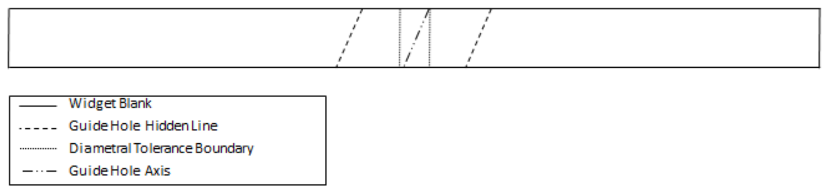


Figure N.25: Translating the extreme center point positions into a diametral zone.

Before this can be entered as the value of the geometric tolerance, the angularity condition for the feature axis must also be checked. Recall from previously, that in order for the axis to meet this aspect of the tolerance requirement, it must fit within a cylinder that is defined by: the diameter called out in the geometric tolerance and the length of the overall feature. This scenario can be seen in Figure N.26.

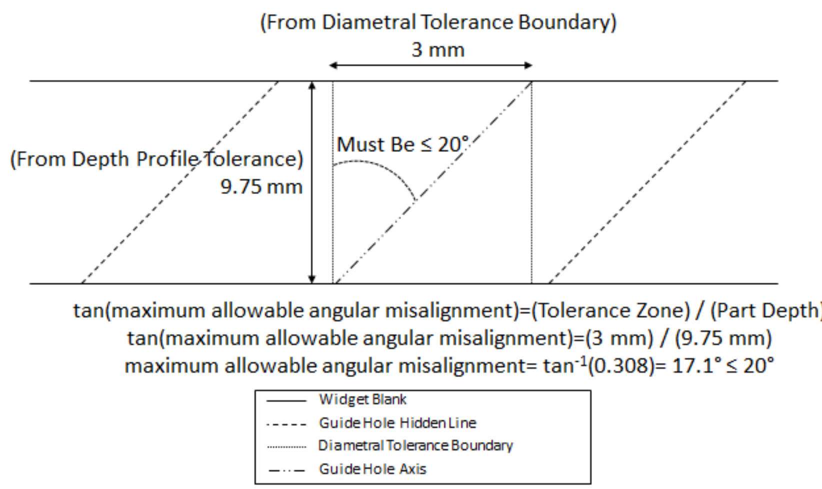


Figure N.26: The maximum angular misalignment for the guide hole.

Again in order to check this requirement, a simple trigonometry problem must be solved. The setup for this problem can be seen in Figure N.27.

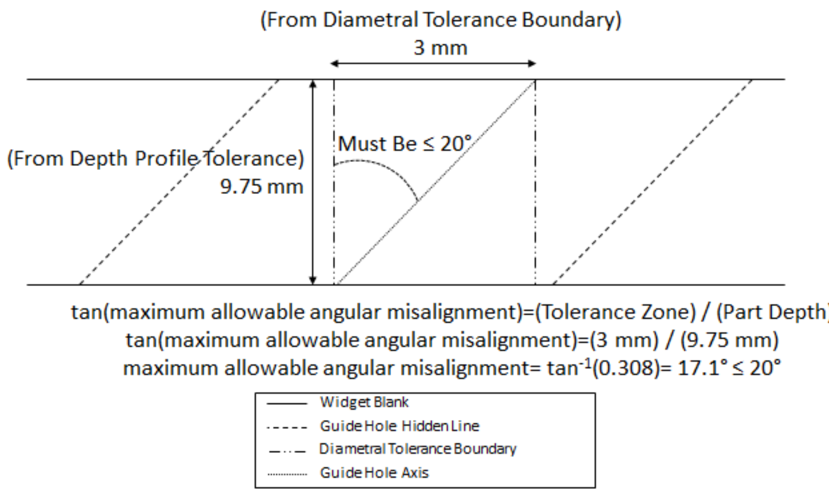


Figure N.27: The trigonometry to check the maximum allowable angular misalignment for the guide hole.

Since the maximum allowable angular misalignment for the hole with a diametral tolerance zone of 3 mm and a depth of 9.75 mm is 17°, the tolerance value will hold for both the position and orientation of the hole.

The true position geometric tolerance can now be added to the drawing as seen in Figure N.28.

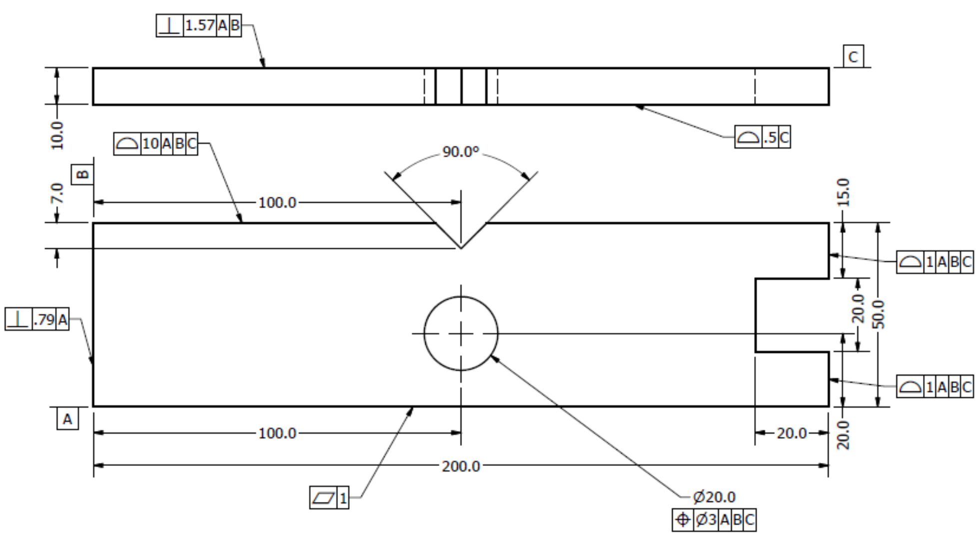


Figure N.28: The true position tolerance added to the 20 mm diameter tolerance.

The guide hole definition is not complete, until a size tolerance is added to the guide hole.

From previously:

The guide hole needs to have a minimum size of 20 mm and a maximum size of 25 mm.

Since the nominal size of the guide hole is 20 mm, the size condition translates into an asymmetric tolerance value of + 5 mm and -0 mm. This can be seen in Figure N.29.

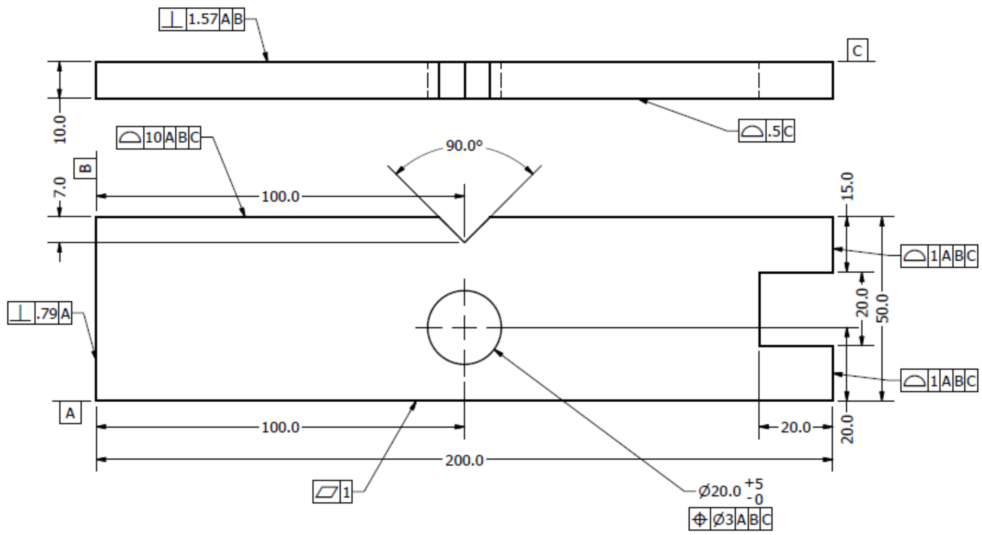


Figure N.29: Adding the dimensional tolerance to the guide hole.

In order to control the orientation and position separately, a tolerance refinement should be used. This is a more complicated topic and will not be covered in this tutorial.

N.2.5 Defining the Non-Functional Features

Since both of the notches are used only to orient the part and are not functional, they do not need rigorous dimensional control. As a result, standard dimensional tolerances will serve just fine in defining the allowable limits for the notches.

Defining the Triangular Notch

According to the process documentation: *The centerline of the triangular notch needs to be located within 5 mm of the horizontal center of the part. The angular shape of the notch* *may vary up to 4°. The depth of the notch may vary 2 mm in either direction.*

Since the notch needs to be located within 5 mm of the center, a symmetric, bi-lateral dimensional tolerance of 5 mm will suffice to define the locational variation. This can be seen in Figure N.30.

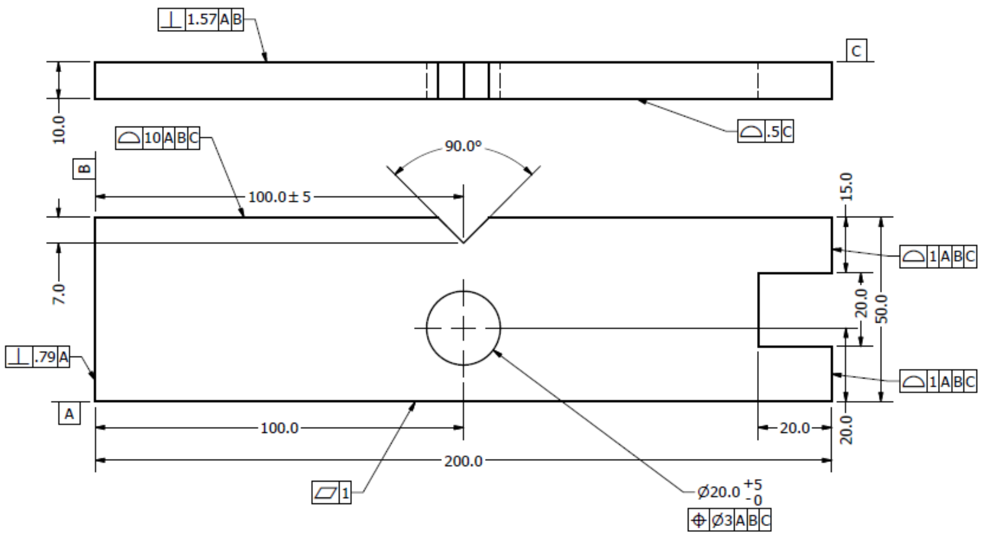


Figure N.30: Defining the locational variation.

The next defining feature of the notch is the angle of the cut. According to the documentation the 90° cut may vary 4°. Another symmetric, bi-lateral dimensional tolerance of 4° will fully define the variation as seen in Figure N.31.

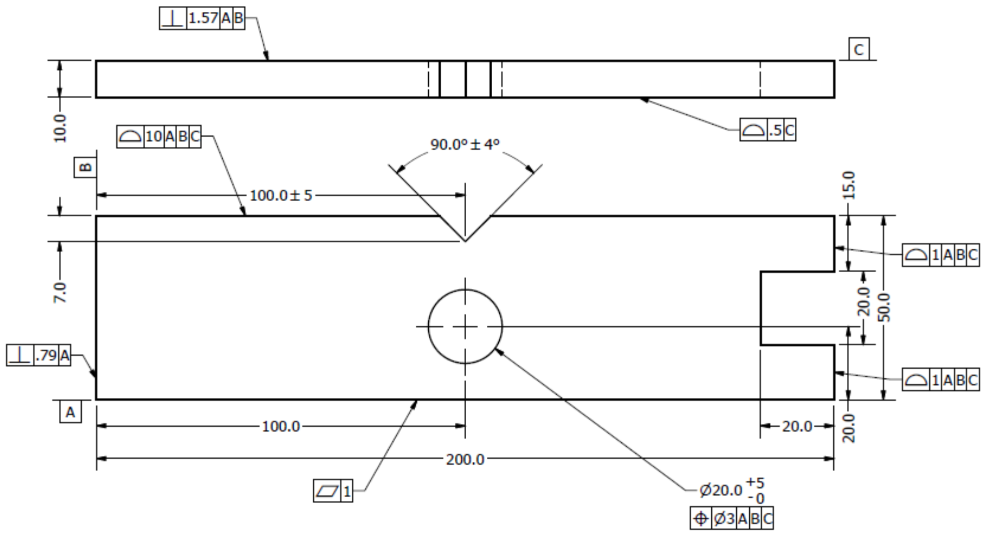


Figure N.31: Defining the allowable angular variation of the cut.

The final defining feature of the triangular notch is the depth of the cut. Again according to the documentation, the depth of the cut may vary 2 mm. This can be controlled through another symmetric, bi-lateral dimensional tolerance as seen in Figure N.32.

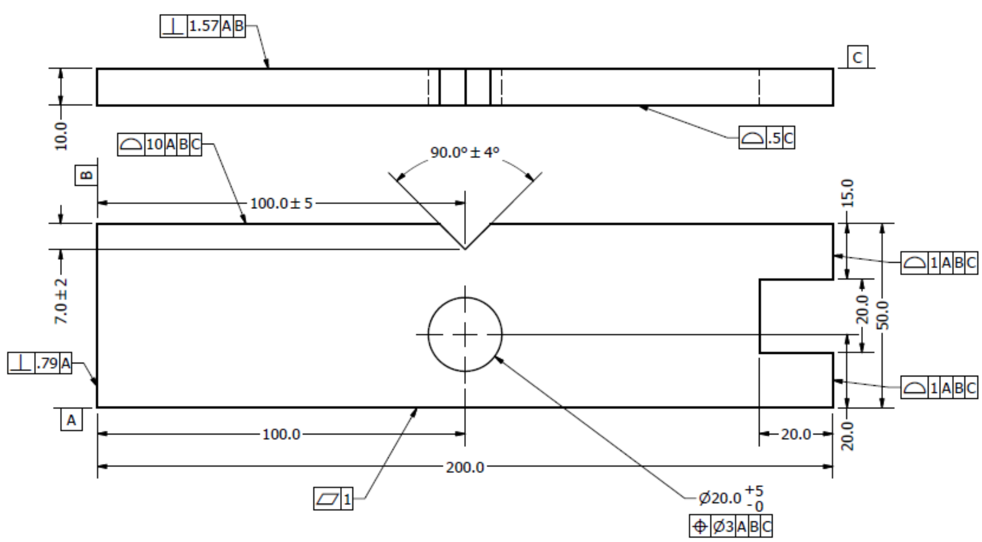


Figure N.32: Defining the allowable variation for the depth of the cut.

Defining the Rectangular Notch

The final feature to define is the rectangular notch. According to the process documentation: *The rectangular notch needs to be located between 10 mm and 20 mm from the top of the part. The depth and height of the notch may vary 2 mm in either direction.*

These variational requirements can again all be controlled with symmetric, bi-lateral dimensional tolerances, because the feature has no rigorous requirements for its function. The requirement of the 10 mm location and 20 mm location can be translated into a tolerance boundary of ±5mm and the size requirements of the notch can be translated into ±2mm. The final drawing can be seen in Figure N.33.

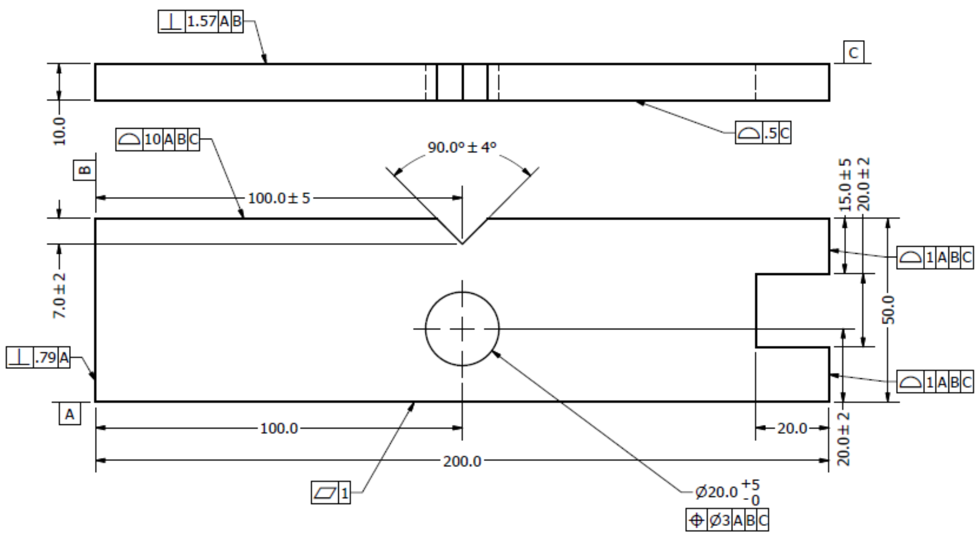


Figure N.33: Defining the variation for the rectangular notch.

APPENDIX O

LAB UNIT TWO - ASSIGNMENT: USING GD&T TO EXPRESS DESIGN INTENT

Your company, Storage Compartments Unlimited, is in the process of finding a supplier to produce lids for one of your storage compartments. It is your responsibility to create an engineering drawing with an appropriate tolerance scheme, so the supplier can create the lids as accurately as possible. You have been given the engineering drawing of the lid with only basic dimensions included. This can be seen in Figure O.1

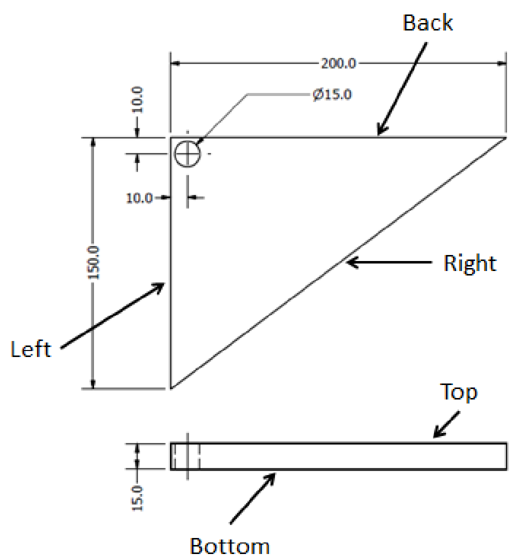


Figure O.1: A drawing of the compartment lid with basic dimensions. Labels have been added to the sides of the lid for easy reference.

O.0.6 Investigation

Before attempting to annotate the drawing with the appropriate tolerances, you decide to do some investigation to see how the lid is assembled.

Schematic

While asking around the office, you locate the following schematic explaining how the product is assembled and works:

1. During assembly, the guide hole on the lid is aligned with the extruded cylinder on the base, as seen in Figure O.2.

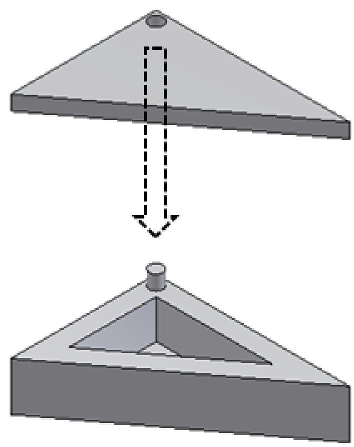


Figure O.2: Step one of the assembly process.

2. Once the lid is in place on the base, it is then attached to the container base. This can be seen in Figure O.3.

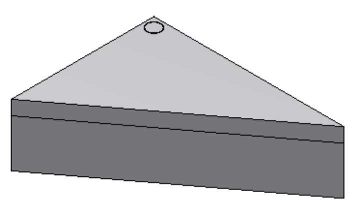


Figure O.3: Step two of the assembly process.

3. After assembly, the lid rotates on its guide post to allow for storage access, see Figure O.4.

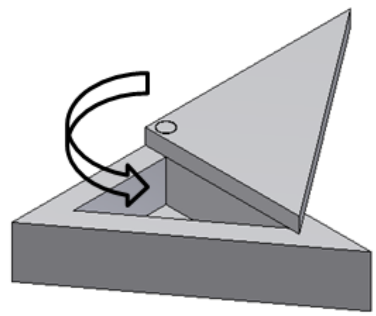


Figure O.4: Storage compartment operation.

Documentation

NOTE: THE FOLLOWING LIST OF REQUIREMENTS ARE USED TO ANSWER THE QUESTIONS IN SECTION O.1, ANNOTATING THE ENGINEERING DRAWING.

After a visit to the manufacturing floor, you discover documentation about the manufacturing process and the requirements for the storage container lids.

The list of requirements for the lid include:

1. The interface between the lid and the container base must not have any gaps larger than 0.5 mm to ensure a tight seal between the two components. The compartment base has been manufactured flat within a 0.01 mm surface finish (since this is an order of magnitude greater, it can be assumed perfectly flat). If this interface is not met, then the storage compartment will not function properly.

2. For aesthetic appearances, the left side surface must remain perpendicular to the bottom face within 0.5°.

3. Again for aesthetic appearances, the back surface must remain perpendicular within 0.5° to both the bottom face and the left side surface.

4. To ensure proper closure between the lid and the base, the right surface must not

vary more than 2 mm in either direction from its nominal (or perfect) position when compared to the left surface, back surface, and bottom surface.

5. The height of the lid is not a feature that needs rigorous control, but it must remain within 1 mm in either direction of the bottom surface.

6. In order for the guide hole to function properly:

   (a) The guide hole needs be positioned from the left surface and the bottom surface somewhere within a diameter of 2 mm located around its perfect location.

   (b) The guide hole needs to have a minimum size of 15 mm and a maximum size of 16 mm.

   (c) The axis of the guide hole needs to have a maximum angular misalignment of 20° with respect to the back surface.

O.1 Annotating the Engineering Drawing

O.1.1 Assigning the Datum Scheme

The first step in annotating an engineering drawing is to assign a datum scheme, which will be used as reference baselines for the part. This drawing will need three datums assigned, the ***-A-*** datum, ***-B-*** datum, and ***-C-*** datum. For clarity, choose the ***-A-*** datum, ***-B-*** datum, and ***-C-*** datum to match the primary, secondary and tertiary datums. Record the requirement numbers (i.e. 1, 6b, 4, etc.) below that are used to choose the datum scheme and add these datums to the attached drawing in Figure O.5.

Datum Scheme Justification

1. Primary Datum Requirement Number (i.e. 1, 6b, 4, etc.):

2. Secondary Datum Requirement Number (i.e. 1, 6b, 4, etc.):

3. Tertiary Datum Requirement Number (i.e. 1, 6b, 4, etc.):

O.1.2 Defining the Datum Scheme Relationship

In a Cartesian coordinate system, it is implied that the XY, YZ, and XZ plane are all mutually perpendicular and 90° to one another. This is not true with datum schemes, and the relationships must be defined.

O.1.3 Controlling the Primary Datum

The first step is to define the form of the primary datum. Record the requirement number (i.e. 1, 6b, 4, etc.) below that is used to define this feature and add the tolerance to the attached drawing, Figure O.5.

Primary Datum Control Justification

4. Requirement Number (i.e. 1, 6b, 4, etc.):

5. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

6. Tolerance Boundary Value:

7. Datum References (i.e. none, A, B, etc.):

O.1.4 Defining the Secondary Datum

The next step, is to relate the secondary datum to the primary datum. Record the requirement number (i.e. 1, 6b, 4, etc.) below that is used to define this feature and add the tolerance to the attached drawing, Figure O.5.

Secondary Datum Control Justification

8. Requirement Number (i.e. 1, 6b, 4, etc.):

9. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

10. Tolerance Boundary Value:

11. Datum References (i.e. none, A, B, etc.):

O.1.5 Defining the Tertiary Datum

The next step is to relate the tertiary datum to the primary datum and the secondary datum. Record the requirement number (i.e. 1, 6b, 4, etc.) below that is used to define this feature and add the tolerance to the attached drawing, Figure O.5.

Secondary Datum Control Justification

12. Requirement Number (i.e. 1, 6b, 4, etc.):

13. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

14. Tolerance Boundary Value:

15. Datum References (i.e. none, A, B, etc.):

O.1.6 Defining the Remaining Surfaces

Now that the datum scheme has been established and defined, it is time to tolerance the remaining surfaces. Complete the justification and add the tolerances to Figure O.5.

Right Surface Control Justification

16. Requirement Number (i.e. 1, 6b, 4, etc.):

17. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

18. Tolerance Boundary Value:

19. Datum References (i.e. none, A, B, etc.):

Top Surface Control Justification

20. Requirement Number (i.e. 1, 6b, 4, etc.):

21. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

22. Tolerance Boundary Value:

23. Datum References (i.e. none, A, B, etc.):

O.1.7 Controlling the Guide Hole

The last feature to be controlled is the guide hole. The guide hole needs position, angularity, and size controlled. Complete the guide hole justification and add the appropriate tolerances to Figure O.5.

Controlling Guide Hole Position and Angularity

24. Requirement Number (i.e. 1, 6b, 4, etc.):

25. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

26. Tolerance Boundary Value:

27. Datum References (i.e. none, A, B, etc.):

28. Maximum axis misalignment with chosen tolerance:

Controlling Guide Hole Size

29. Requirement Number (i.e. 1, 6b, 4, etc.):

30. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

31. Tolerance Boundary Value:

32. Datum References (i.e. none, A, B, etc.):

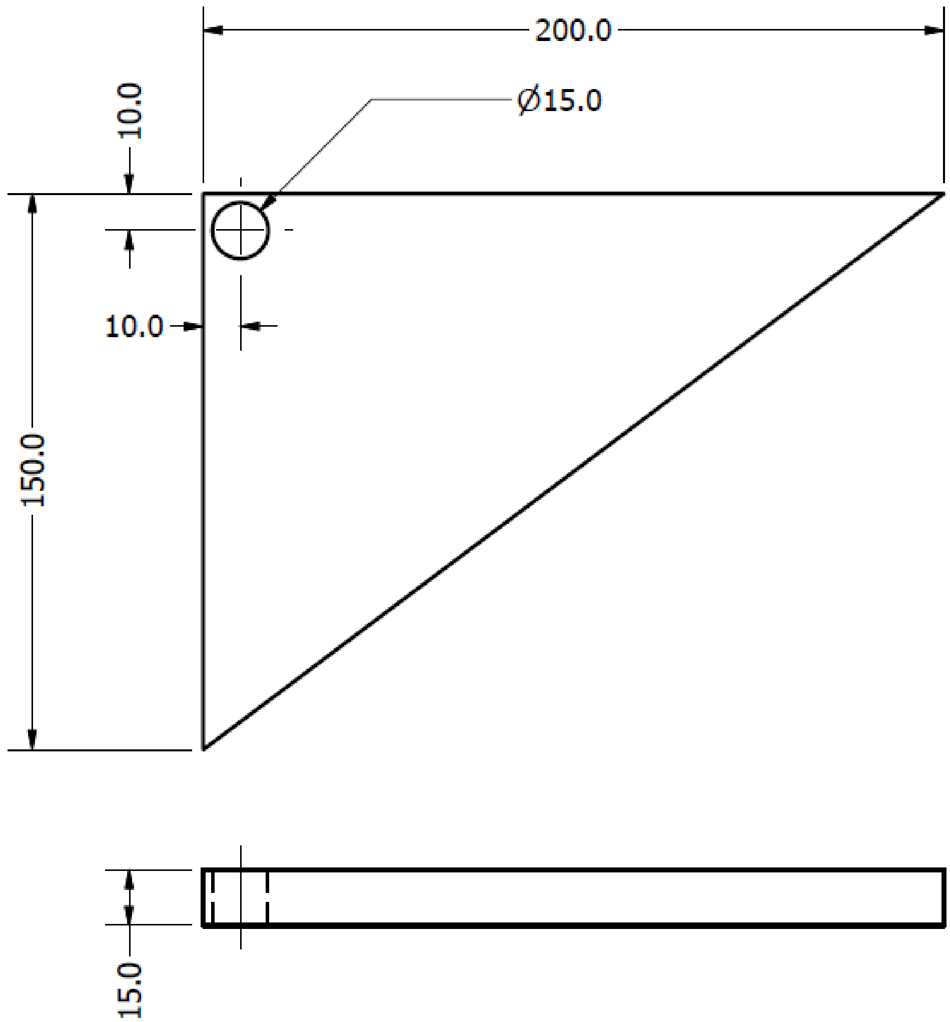


Figure O.5: Complete this drawing with datums and the appropriate tolerances.

APPENDIX P

LAB UNIT TWO - ASSIGNMENT: USING GD&T TO EXPRESS DESIGN INTENT

Your company, Storage Compartments Unlimited, is in the process of finding a supplier to produce lids for one of your storage compartments. It is your responsibility to create an engineering drawing with an appropriate tolerance scheme, so the supplier can create the lids as accurately as possible. You have been given the engineering drawing of the lid with only basic dimensions included. This can be seen in Figure O.1

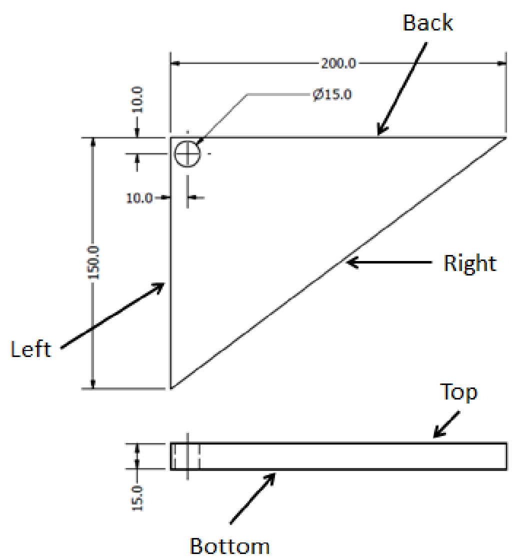


Figure P.1: A drawing of the compartment lid with basic dimensions. Labels have been added to the sides of the lid for easy reference.

P.0.8 Investigation

Before attempting to annotate the drawing with the appropriate tolerances, you decide to do some investigation to see how the lid is assembled.

Schematic

While asking around the office, you locate the following schematic explaining how the product is assembled and works:

1. During assembly, the guide hole on the lid is aligned with the extruded cylinder on the base, as seen in Figure O.2.

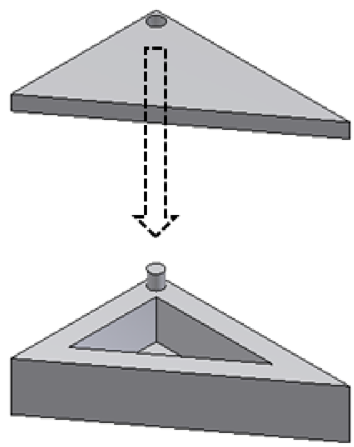


Figure P.2: Step one of the assembly process.

2. Once the lid is in place on the base, it is then attached to the container base. This can be seen in Figure O.3.

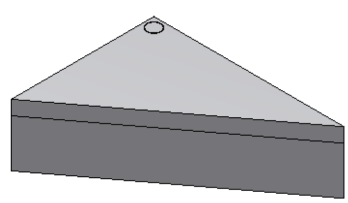


Figure P.3: Step two of the assembly process.

3. After assembly, the lid rotates on its guide post to allow for storage access, see Figure O.4.

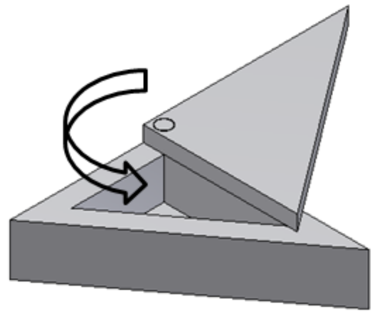


Figure P.4: Storage compartment operation.

Documentation

NOTE: THE FOLLOWING LIST OF REQUIREMENTS ARE USED TO ANSWER THE QUESTIONS IN SECTION O.1, ANNOTATING THE ENGINEERING DRAWING.

After a visit to the manufacturing floor, you discover documentation about the manufacturing process and the requirements for the storage container lids.

The list of requirements for the lid include:

1. The interface between the lid and the container base must not have any gaps larger than 0.5 mm to ensure a tight seal between the two components. The compartment base has been manufactured flat within a 0.01 mm surface finish (since this is an order of magnitude greater, it can be assumed perfectly flat). If this interface is not met, then the storage compartment will not function properly.

2. For aesthetic appearances, the left side surface must remain perpendicular to the bottom face within 0.5°.

3. Again for aesthetic appearances, the back surface must remain perpendicular within 0.5° to both the bottom face and the left side surface.

4. To ensure proper closure between the lid and the base, the right surface must not

vary more than 2 mm in either direction from its nominal (or perfect) position when compared to the left surface, back surface, and bottom surface.

5. The height of the lid is not a feature that needs rigorous control, but it must remain within 1 mm in either direction of the bottom surface.

6. In order for the guide hole to function properly:

   (a) The guide hole needs be positioned from the left surface and the bottom surface somewhere within a diameter of 2 mm located around its perfect location.

   (b) The guide hole needs to have a minimum size of 15 mm and a maximum size of 16 mm.

   (c) The axis of the guide hole needs to have a maximum angular misalignment of 20° with respect to the back surface.

P.1 Annotating the Engineering Drawing

P.1.1 Assigning the Datum Scheme

The first step in annotating an engineering drawing is to assign a datum scheme, which will be used as reference baselines for the part. This drawing will need three datums assigned, the ***-A-*** datum, ***-B-*** datum, and ***-C-*** datum. For clarity, choose the ***-A-*** datum, ***-B-*** datum, and ***-C-*** datum to match the primary, secondary and tertiary datums. Record the requirement numbers (i.e. 1, 6b, 4, etc.) below that are used to choose the datum scheme and add these datums to the attached drawing in Figure O.5.

Datum Scheme Justification

1. Primary Datum Requirement Number (i.e. 1, 6b, 4, etc.): ***1***

2. Secondary Datum Requirement Number (i.e. 1, 6b, 4, etc.): ***2***

3. Tertiary Datum Requirement Number (i.e. 1, 6b, 4, etc.): ***3***

P.1.2 Defining the Datum Scheme Relationship

In a Cartesian coordinate system, it is implied that the XY, YZ, and XZ plane are all mutually perpendicular and 90° to one another. This is not true with datum schemes, and the relationships must be defined.

P.1.3 Controlling the Primary Datum

The first step is to define the form of the primary datum. Record the requirement number (i.e. 1, 6b, 4, etc.) below that is used to define this feature and add the tolerance to the attached drawing, Figure O.5.

Primary Datum Control Justification

4. Requirement Number (i.e. 1, 6b, 4, etc.): ***1 - See attached drawing for remaining***

5. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

6. Tolerance Boundary Value:

7. Datum References (i.e. none, A, B, etc.):

P.1.4 Defining the Secondary Datum

The next step, is to relate the secondary datum to the primary datum. Record the requirement number (i.e. 1, 6b, 4, etc.) below that is used to define this feature and add the tolerance to the attached drawing, Figure O.5.

Secondary Datum Control Justification

8. Requirement Number (i.e. 1, 6b, 4, etc.): ***2- See attached drawing for remaining***

9. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

10. Tolerance Boundary Value:

11. Datum References (i.e. none, A, B, etc.):

P.1.5 Defining the Tertiary Datum

The next step is to relate the tertiary datum to the primary datum and the secondary datum. Record the requirement number (i.e. 1, 6b, 4, etc.) below that is used to define this feature and add the tolerance to the attached drawing, Figure O.5.

Secondary Datum Control Justification

12. Requirement Number (i.e. 1, 6b, 4, etc.): ***3 - See attached drawing for remaining***

13. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

14. Tolerance Boundary Value:

15. Datum References (i.e. none, A, B, etc.):

P.1.6 Defining the Remaining Surfaces

Now that the datum scheme has been established and defined, it is time to tolerance the remaining surfaces. Complete the justification and add the tolerances to Figure O.5.

Right Surface Control Justification

16. Requirement Number (i.e. 1, 6b, 4, etc.): ***4 - See attached drawing for remaining***

17. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

18. Tolerance Boundary Value:

19. Datum References (i.e. none, A, B, etc.):

Top Surface Control Justification

20. Requirement Number (i.e. 1, 6b, 4, etc.): ***5 - See attached drawing for remaining***

21. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

22. Tolerance Boundary Value:

23. Datum References (i.e. none, A, B, etc.):

P.1.7 Controlling the Guide Hole

The last feature to be controlled is the guide hole. The guide hole needs position, angularity, and size controlled. Complete the guide hole justification and add the appropriate tolerances to Figure O.5.

Controlling Guide Hole Position and Angularity

24. Requirement Number (i.e. 1, 6b, 4, etc.): ***6a and 6c - See attached drawing for remaining***

25. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

26. Tolerance Boundary Value:

27. Datum References (i.e. none, A, B, etc.):

28. Maximum axis misalignment with chosen tolerance:

Controlling Guide Hole Size

29. Requirement Number (i.e. 1, 6b, 4, etc.): ***6b- See attached drawing for remaining***

30. Tolerance Type Added (i.e. Dimensional, True Position, Perpendicularity, etc.):

31. Tolerance Boundary Value:

32. Datum References (i.e. none, A, B, etc.):

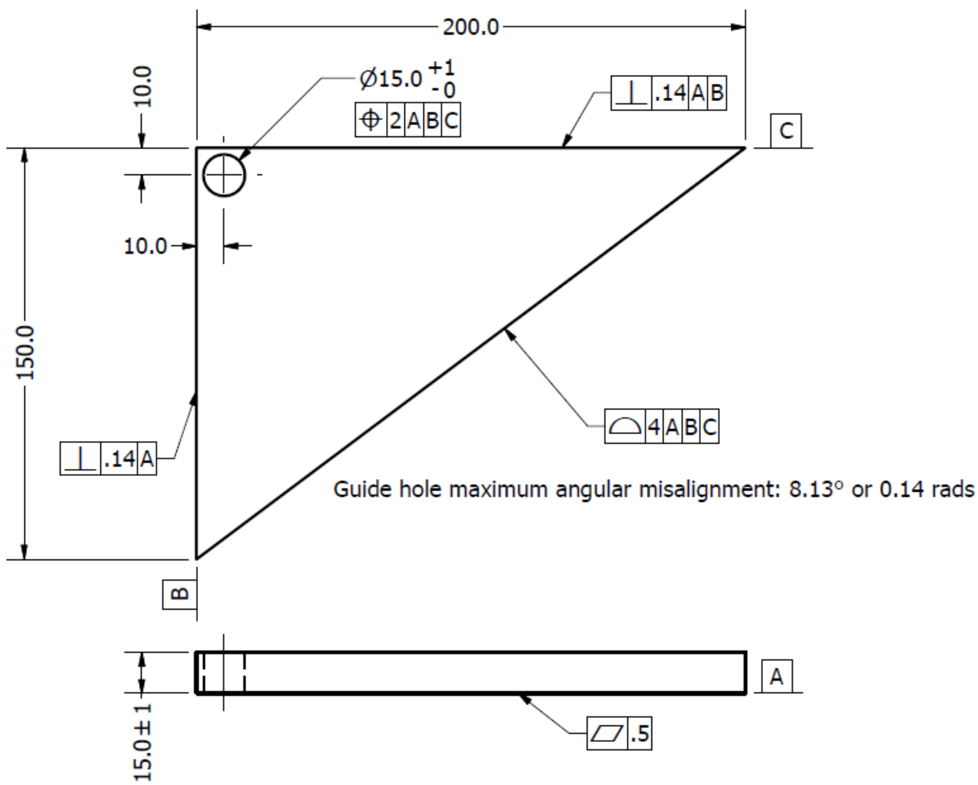


Figure P.5: Complete drawing of Figure O.5.

APPENDIX Q

SUPPLEMENTARY FILES

The Autodesk Inventor parts, Microsoft Excel files, and additonal reference material may be found in the supplemental file supplementary_files.zip.

REFERENCES

[1] M. Henry, "Interview," 2010, statistical Tolerance Analysis Team Leader - Caterpillar, Inc.

[2] T. A. S. of Mechanical Engineers, Dimensioning and Tolerancing Standard ASME Y14.5M - 1994. New York, New York: The American Society of Mechanical Engineers, 1995.

[3] Efunda, "Geometric Dimensioning and Tolerancing: Why?" available at http://www.efunda.com/designstandards/gdt/introduction.cfm.

[4] A. Neumann, Geometric Dimensioning and Tolerancing Workbook. Dearborn, MI: Society of Manufacturing Engineers, 1995.

[5] J. D. Meadows, Tolerance Stack-Up Analysis. Hendersonville, TN: James D. Meadows & Associates, Inc., 2001.

[6] Sigmetrix, "Professional services - training," available at http://www.sigmetrix.com/CETOL6_sigma_training.htm.

[7] Sigmetrix, "Client List," available at http://www.sigmetrix.com/CETOL6_clients.htm.

[8] U. N. . W. Report, "Best Engineering Schools Specialty Rankings: Mechanical," available at http://grad-schools.usnews.rankingsandreviews.com/best-graduate-schools/top-engineering-schools/mechanical-engineering.

[9] B. University of California, "Mechanical Engineering Courses," available at http://sis.berkeley.edu/catalog/gcc_list_crse_req?p_dept_name=Mechanical+Engineering&p_dept_cd=MEC+ENG.

[10] C. I. of Technology, "Mechanical Engineering Courses," available at http://www.me.caltech.edu/academics/courses.html.

[11] G. I. of Technology, "Mechanical Engineering Courses," available at http://www.me.gatech.edu/undergraduate/degrees_bsme_curr.shtml.

[12] P. University, "Mechanical Engineering Courses," available at http://www.princeton.edu/mae/undergraduate/courses/.

[13] M. I. of Technology, “Mechanical Engineering Courses,” available at http://student.mit.edu/catalog/m2a.html.

[14] S. University, “Mechanical Engineering Courses,” available at http://scpd.stanford.edu/search/publicCourseSearchDetails.do?method=load&courseId=12592.

[15] P. University, “Mechanical Engineering Courses,” available at https://esa-oas-prod.itap.purdue.edu:9011/prod/bwckctlg.p_display_courses.

[16] U. of Michigan, “Mechanical Engineering Courses,” available at https://me-web2.engin.umich.edu/courses/.

[17] U. of Illinois, “Mechanical Engineering Courses,” available at http://courses.illinois.edu/cis/2010/fall/catalog/ME/index.html?skinId=2169.

[18] C. University, “Mechanical Engineering Courses,” available at http://courses.cuinfo.cornell.edu/CoSdetail.php?college=ENG&number=5200&prefix=MAE&title=Dimensional+Tolerancing+in+Mechanical+Design.

[19] A. S. for Engineering Education, “The Organization-Who We Are,” available at http://www.asee.org/about-us/the-organization.

[20] L. E. Carlson and J. C. Trabert, “Virtual cad parts to enhance learning of geometric dimensioning and tolerancing,” in Proceedings of the 2005 American Society for Engineering Education Annual Conference & Exposition. American Society for Engineering Education, 2005.

[21] Immersion, “MicroScribe G2 Desktop Digitizing Systems User’s Guide.”

[22] ReverseEngineering.com, “HIGHRES CAD Integrated 3D Reverse Engineering Plug-ins,” specification Sheet.

[23] Starrett, “Granite Surface Plates,” available at http://www.starrett.com/pages/860_granite_surface_plates.cfm.

[24] Starrett, “Frequently Asked Questions,” available at http://www.tru-stone.com/pages/faq.asp.

[25] McMaster-Carr, “Precision Ground Aluminum Blank Data Sheet,” available at http://www.mcmaster.com/#9057k34/=9c3rcz.

[26] T. Wholers, Wholers Report 2005. Fort Collins, Colorado: Wholers Associate, Inc., 2005.

[27] T. Grimm, “3d printer dimensional accuracy,” Time-Compression Technologies, vol. 13, no. 5, pp. 1–4, September/October 2005.

[28] StrataSys, “Dimension 1200 SST User Manual.”

[29] D. Systems, “InVision SR 3-D Printer Data Sheet.”

[30] Z. Corp, "ZPrinter 310 Plus Hardware Manual."

[31] Autodesk, "Autodesk Inventor System Requirements," available at http://usa.autodesk.com/adsk/servlet/pc/index?siteID=123112&id=13727144.

[32] E. M. Raisor, Engineering Graphics Principles with Geometric Dimensioning and Tolerancing. Mission, KS: Schroff Development Corp, 2002.

[33] I. Zeid, Mastering CAD/CAM. New York, NY: McGraw-Hill Science/Engineering/Math, 2004.

[34] J. Leake and J. L. Borgerson, Engineering Design Graphics: Sketching, Modeling, and Visualization. Hoboken, NJ: John Wiley & Sons Inc, 2008.